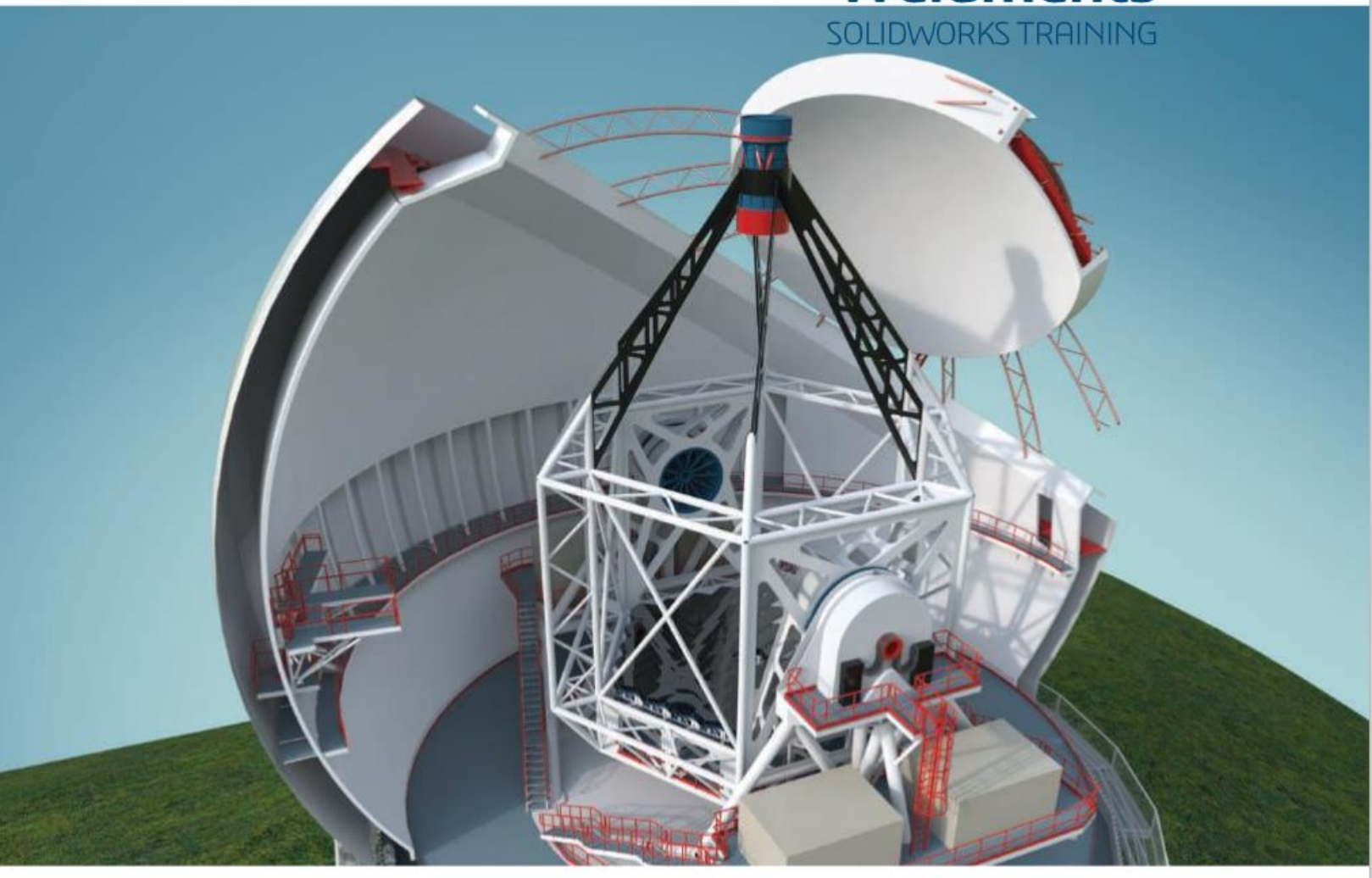
DASSAULT SYSTEMES

SOLIDWORKS

Weldments

SOLIDWORKS TRAINING



ENG

SOLIDWORKS®

Weldments

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Introduction

About This Course

The goal of this course is to teach you how to build weldment parts using SOLIDWORKS mechanical design automation software.

The weldment application in the SOLIDWORKS software is so robust and feature rich that it is impractical to cover every minute detail and still have the course be a reasonable length. Therefore, the focus of this course is on the fundamental skills and concepts central to successfully building weldment parts. You should view the training course manual as a supplement to, not a replacement for, the system documentation and on-line help. Once you have developed a good foundation in basic skills, you can refer to the on-line help for information on less frequently used command options.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Completed the course *SOLIDWORKS Essentials*.
- Experience with the Windows® operating system.

Course Length

The recommended minimum length for this course is one day.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. A process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced SOLIDWORKS instructor. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated "live" by the instructor.

Laboratory Exercises

Laboratory exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course. They are designed to represent typical design and modeling situations while being modest enough to be completed during class time. You should note that many students work at different paces. Therefore, we have included more lab exercises than you can reasonably expect to complete during the course. This ensures that even the fastest student will not run out of exercises.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that compliments this objective.

Conventions Used in this Book

This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, **Features > Extruded Cut** means click the **Extruded Cut** icon on the **Features** tab of the CommandManager.
Typewriter	Feature names and file names appear in this style. For example, Sketch1.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

About the Training Files

A complete set of the various files used throughout this course can be downloaded from the SOLIDWORKS website, **www.solidworks.com**. Click on the link for **Support**, then **Training**, then **Training Files**, then **SOLIDWORKS Training Files**. Select the link for the desired file set. There may be more than one version of each file set available.

Direct URL:

www.solidworks.com/trainingfilessolidworks

The files are supplied in signed, self-extracting executable packages. The default extraction location for the training files will be C:\SOLIDWORKS Training Files.

The files are organized by lesson number. The Case Study folder within each lesson contains the files your instructor uses while presenting the lessons. The Exercises folder contains any files that are required for doing the laboratory exercises. Completed lesson files are also included and the files names are followed by "_&" to differentiate them.

Training Templates

A set of training templates are used throughout the course to standardize the settings for the new files that are created. These templates are also supplied in signed, self-extracting executable packages. They can be downloaded from the same website address as the Training Files and use the same extraction location.

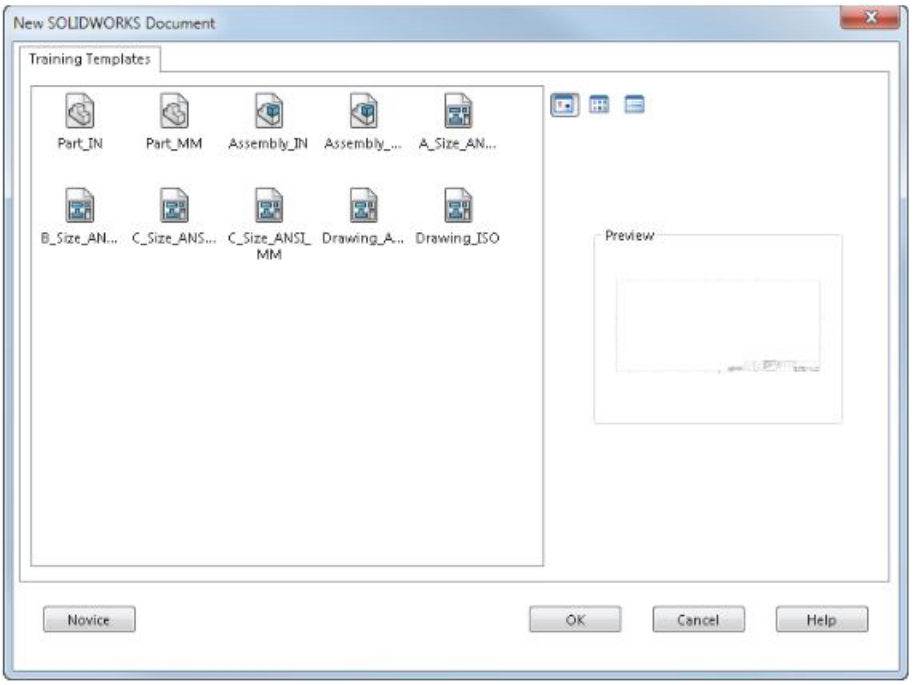
These templates are used for all SOLIDWORKS core courses.

To access the training templates in SOLIDWORKS:

1. Click **Tools**, **Options**, **System Options**, **File Locations**.
2. Choose **Document Templates** from the drop down menu.
3. Click **Add** and browse to the Training Templates folder.
4. Click **OK** and **Yes** on each of the messages.

Accessing Training Templates

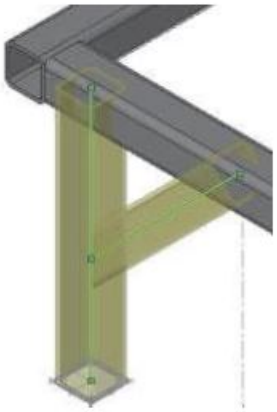
Once the file location has been added, click the **Advanced** button and the **Training Templates** tab in the **New SOLIDWORKS Document** dialog can be used.



Windows

The screen shots in this manual were made using the SOLIDWORKS software running a mixture of Windows® 7 and Windows 10. You may notice slight differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Use of Color



The SOLIDWORKS user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the SOLIDWORKS software. To take maximum advantage of this, the training manuals are printed in full color.

Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of an operation in a different color, even though by default, the SOLIDWORKS software would not display the results in that way.

Graphics and Graphics Cards

The SOLIDWORKS software sets a new standard with best-in-class graphics. The combination of a highly reflective material and the realism of **RealView Graphics** is an effective tool for evaluating the quality of advanced part models and surfaces.

RealView Graphics is hardware (graphics card) support of advanced shading in real time. For example, if you rotate a part, it retains its rendered appearance throughout the rotation.

Color Schemes

Out of the box, the SOLIDWORKS software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

We have not used the same color scheme for every case study and exercise because some colors are more visible and clear than others when used with different colored parts.

In addition, we have changed the viewport background to plain white so that the illustrations reproduce better on white paper.

As a result, because the color settings on your computer may be different than the ones used by the authors of this book, the images you see on your screen may not exactly match those in the book.

User Interface Appearance

Throughout the development of the software, there have been some cosmetic User Interface changes, intended to improve visibility, that do not affect the function of the software. As a policy, dialog images in the manuals which exhibit no functional change from the previous version are not replaced. As such, you may see a mixture of current and "old" UI dialogs and color schemes.

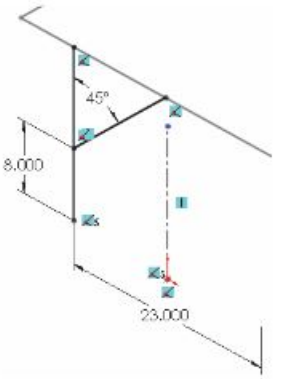
Finding Commands

Not all commands covered in this course are available on a toolbar by default. For example, the **Fillet Bead** command is not on the Weldments toolbar even though it is considered a Weldment feature. If you want to access these commands from the toolbars, they can be added using **Tools, Customize**.

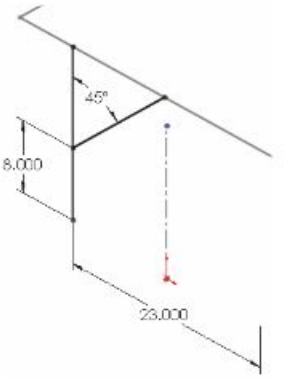
Alternatively, consider using the **Command Search** at the top of the SOLIDWORKS interface to find and access commands that are not readily available.

Displaying Sketch Relations

The SOLIDWORKS software has the capability to automatically display graphic representations of sketch relations as shown in the illustration at the right. These can be turned on and off using the command **View, Hide / Show, Sketch Relations**.



In most of the examples in this book, the graphic display of the sketch relations has been turned off.



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Discover the benefits of the SOLIDWORKS User Group Network (SWUGN). Attend local meetings to hear technical presentations on SOLIDWORKS and related engineering topics, learn about additional SOLIDWORKS products, and network with other users. Groups are led by SOLIDWORKS users just like you. Check out SWUGN.org for more information, including how to find a group in your area.

Lesson 1
Weldment Features

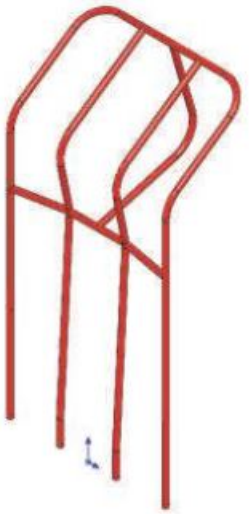
Upon successful completion of this lesson, you will be able to:

- Understand how a Weldment feature affects a part model.
- Create structural members features.
- Download standard structural member profiles.
- Manage corner treatments and trimming of structural members.
- Create gussets and end caps.

Weldments

In general, a weldment is an object made up of several parts welded together. In SOLIDWORKS, a weldment refers to special type of part model containing multiple bodies which can be described with a cut list. Often times these bodies are welded together in production, such as structural members welded together to form a frame.

Even though a weldment model could technically be described as an assembly, working within a multibody part allows for easy manipulation of multiple pieces and minimizes complex file relationships. Specialized weldment commands also automate common functions of working with structural members and frames.



Although primarily targeted for working with structural steel and aluminum, SOLIDWORKS weldments are also commonly used for modeling wood working projects and plastic extrusions.

Note

Details of working with conventional multibody parts are covered in the *SOLIDWORKS Advanced Part Modeling* course.

Weldment Commands

The specialized weldment commands are grouped together on the Weldments tab of the CommandManager. The weldment commands allow you to:

- Insert structural members.
- Use a special-purpose tool for trimming or extending structural members.
- Add gussets, end caps, and weld beads.

Tip

The **Weldments** tab is not available in the CommandManager by default. To show additional tabs in the CommandManager, right-click any visible tab and select from the list of available tabs.

Weldment Feature

A weldment part model will have a **Weldment** feature appearing as the first feature in the FeatureManager design tree. This feature can be added manually from the weldments toolbar, or will automatically be applied if a **Structural Member** feature is used. Adding the weldment feature to a part does the following:

- Enables the specialized weldment commands.
- Replaces the **Solid Bodies** folder with the **Cut list** . This folder is used to organized the bodies of the part and add properties that can be displayed in a cut list table (see *Managing the Cut List* on page 64).
- Configurations are added to allow for the model to be represented in different stages of the manufacturing process. How these configurations are created or if they are created can be controlled with options (see *Weldment Configuration Options* on page 10).
- The **Merge Result** check box is automatically cleared for all subsequent features that use this option. This allows new features to remain as separate discrete bodies by default.
- Custom properties: The weldment feature acts as a holder for custom properties that are to be inherited by each cut list item (see *Adding Cut List Properties* on page 68).

You can insert only one weldment feature per part. It will always appear as the first feature regardless of when you insert it.

Where to Find It

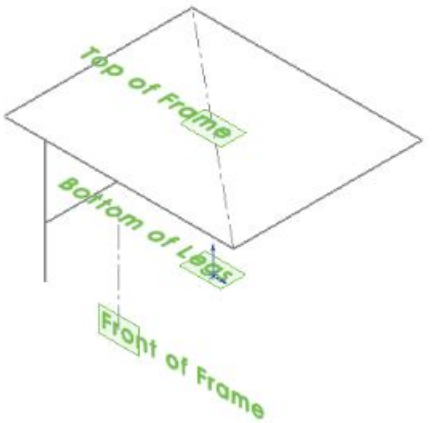
- CommandManager: **Weldments > Weldment**
- Menu: **Insert, Weldments, Weldment**

---

1 Open part.

Open the existing part `Conveyor Frame` from the `Lesson01\Case Study` folder.

This part currently has a single `Default` configuration and includes layout sketches that will be used to create structural members.



2 Weldment feature.

Click **Weldment** .

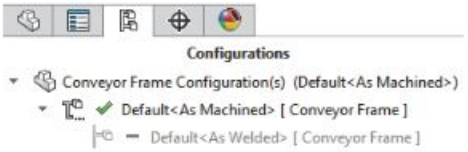
The `Weldment` feature is added to the FeatureManager design tree and configurations are added to the part.

Tip

If you do not insert the weldment feature, one will be added automatically when you insert the first structural member.

Weldment Configuration Options

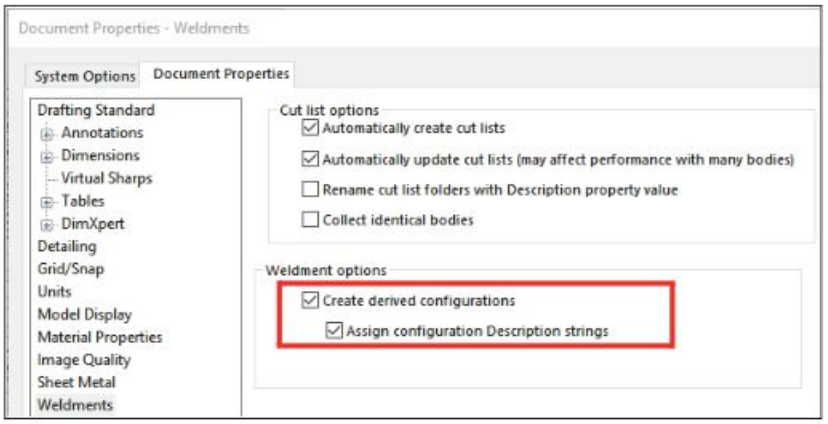
The default behavior when adding the **Weldment** feature includes the creation of a derived configuration and configuration descriptions as follows:



- The description <As Machined> is added to the active configuration.
- A new derived configuration with the same name is added with the description <As Welded>.
- Additionally, any new top level configurations created will automatically have a corresponding <As Welded> derived configuration.

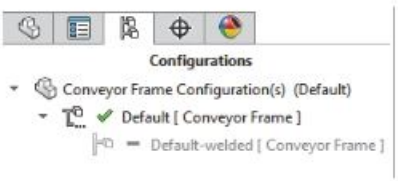
These configurations provide a means for representing the weldment as it will initially be welded and as it will be after post-assembly machining operations.

In **Options** , **Document Properties, Weldments**, settings can be adjusted to modify how these configurations are created.



Clearing the selection for **Assign configuration Description strings** would provide the following behavior:

- A new derived configuration is added with a suffix of "-welded" appended to the configuration name.
- Any subsequent top level configurations created will automatically have a corresponding "-welded" derived configuration.



Clearing the option to **Create derived configurations** will prevent the creation of additional configurations in the part.

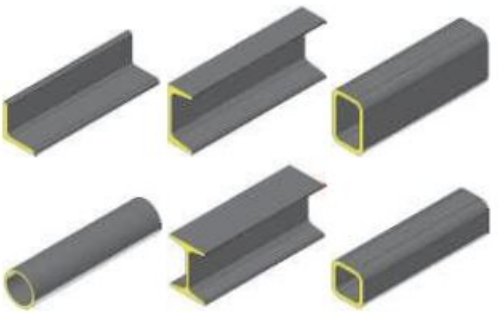
Tip

To set up a standard for how weldment model configurations are created, consider modifying these document properties and saving them to a new part template.

Structural Members

Structural members are generally lengths of structural steel or aluminum tube, pipe, beams, or channels. **Structural Member** *features* are the primary feature for SOLIDWORKS weldment models. They are created by first setting up a layout of lines and arcs in 2D and 3D sketches, then within the **Structural Member** PropertyManager, a structural member profile is selected to sweep along the layout segments. By default, each separate sketch segment will result in a separate solid body in the model, although by using options, this can be modified. Options also exist to modify the corner conditions between the structural member bodies and adjust their orientation and position along the layout.

The structural member profile, or weldment profile, represents a cross section of the structural member that will be created, as shown in yellow at the right.



Only a limited number of structural member profiles are initially installed with SOLIDWORKS to reduce unnecessary data. Complete collections of standard profiles and sizes can be downloaded through SOLIDWORKS Content.

The Default Profiles

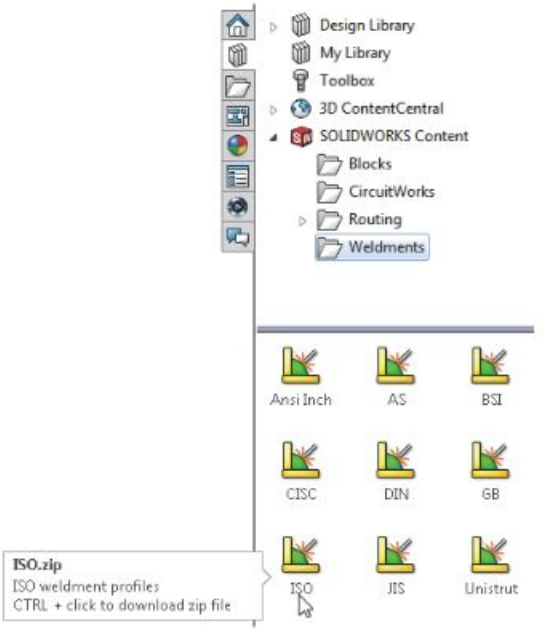
The profiles available by default are shown in the following table.

Standard	Type	Standard	Type
ANSI inch	■ angle iron ■ c channel ■ pipe ■ rectangular tube ■ s section ■ square tube	**ISO**	■ angle iron ■ c channel ■ pipe ■ rectangular tube ■ sb beam ■ square tube

Weldment Profiles from SOLIDWORKS Content

To download a complete set of standard structural member profiles access the **Task Pane**, **Design Library** tab, and select **SOLIDWORKS Content, Weldments**.

Press **Ctrl** and click on the icon representing the standard you want to download. The profiles will be in zip format. The following tables summarize the profile types for each available standard.



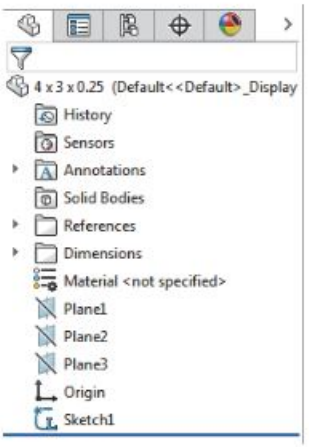
Standard	Type	
ANSI inch	■ AI Channel (standard) ■ AI CS Channel (squared ends) ■ AI I Beam ■ AI I Beam (standard) ■ AI L Angle (rounded ends) ■ AI LS Angle (squared ends) ■ AI Pipe (structural) ■ AI Round Tubing ■ AI T Section ■ AI Tube (rectangular) ■ AI Tube (square) ■ AI Z Section ■ c channel ■ HP Section ■ L Angle ■ M Section	■ MC Channel ■ MT Section ■ Pipe (standard, S40) ■ Pipe (X strong, S80) ■ Pipe (XX strong) ■ s section ■ ST Section ■ Tube (rectangular) ■ Tube (square) ■ W Section ■ WT Section

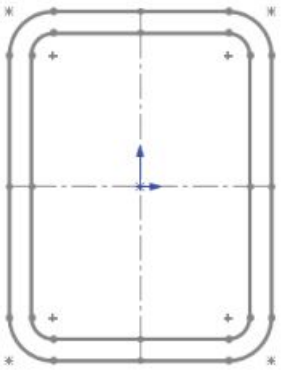
Standard	Type		
AS	■ Circular Hollow Section (C250) AS ■ Circular Hollow Section (C350) AS ■ Equal Angle (New Zealand) AS-NZS ■ Equal Angle AS-NZS ■ Parallel Flange Channel AS-NZS ■ Rectangular Hollow Section (C350) AS ■ Rectangular Hollow Section (C450) AS ■ Square Hollow Section (C350) AS ■ Square Hollow Section (C450) AS ■ Tapered Flange Beam AS-NZS ■ Tapered Flange Channel AS-NZS ■ Tube (Heavy) AS ■ Tube (Light) AS ■ Tube (Medium) AS ■ Unequal Angles AS-NZS ■ Universal Beam AS-NZS ■ Universal Column AS-NZS ■ Welded Beam AS-NZS ■ Welded Column AS-NZS		
BSI	■ CHS Pipe ■ RSA Angle ■ RSC Channel ■ RSJ Beam ■ Tube (rectangular) ■ Tube (square)		■ UB Beam ■ UBP Beam ■ UBT Tee ■ UC Beam ■ UCT Tee
CISC	■ c channel ■ HP Section ■ HS Pipe ■ L Angle ■ M Section	■ MC Channel ■ s section ■ Tube (rectangular) ■ Tube (square)	■ W Section ■ WT Section ■ WWF Section ■ WWT Section
DIN	■ c Channel ■ DIL Beam ■ HD Beam ■ HE Beam ■ HL Beam ■ HP Beam ■ HX Beam	■ IPE Beam ■ IPEA Beam ■ IPEO Beam ■ IPER Beam ■ IPEV Beam ■ IPN Beam	■ L Angle ■ M Beam ■ S Beam ■ U Channel ■ UPN Channel ■ W Beam
GB	■ Channel ■ Double T ■ Hexagonal Bar	■ L Angle (equal) ■ L Angle (unequal)	■ Round Bar ■ Square Bar

Standard	Type		
ISO	■ c channel ■ Circular Tube ■ L Angle (equal)	■ L Angle (unequal) ■ SB Beam ■ SC Beam	■ T Section ■ Tube (rectangular) ■ Tube (square)
JIS	■ Channel ■ H Section	■ I Section ■ L Angle (equal)	■ L Angle (unequal)
Unistrut	■ Aluminum ■ Fiberglass	■ Steel 114	■ Steel 1316 ■ Steel 158

What is a Structural Member Profile?

A structural member profile is simply a 2D, closed contour sketch saved as a library feature part (*.sldlfp). The file locations of structural member profiles must be specified in SOLIDWORKS System Options. Custom profiles can be created to allow for a wide variety of uses for weldment model types. For more information on custom weldment profiles see *Custom Structural Member Profiles* on page 75.





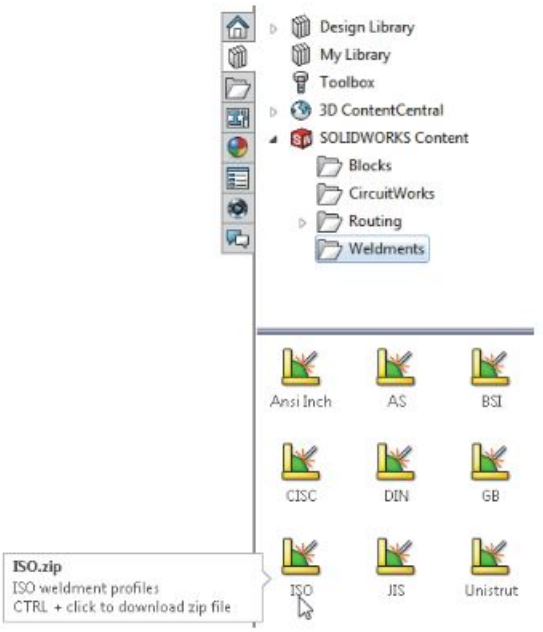
3 Download ISO standard profiles.

Click the **Design Library** tab.

Expand **SOLIDWORKS Content** and select the **Weldments** folder.

Press **Ctrl** and click on the **ISO** icon to download the zip file.

Save the zip file to a location on the local disk, such as the Desktop.



4 Extract files.

Once the download is complete, extract the zip file to the Weldment Profiles folder provided in the training files directory for this course.

Important

The Zip extraction adds a folder named ISO. **Rename** this folder to ISO_Training.

Note

The Weldment Profiles folder includes a portion of the ANSI Inch standard which will be used in later lessons.

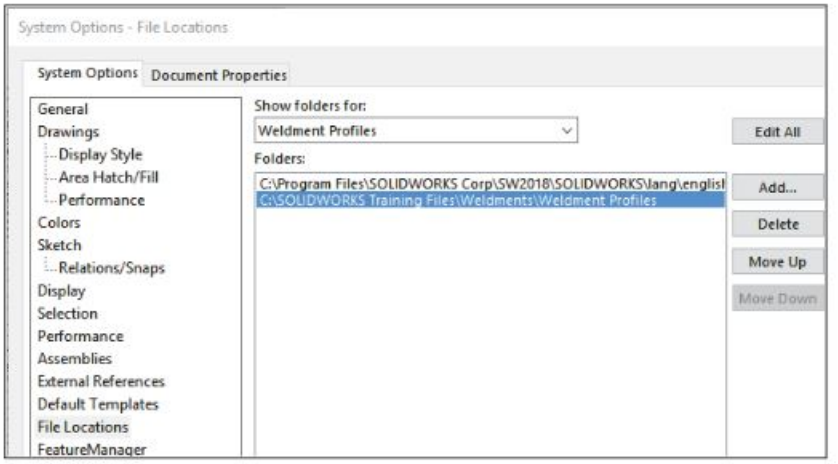
5 File Locations.

In order for SOLIDWORKS to recognize the downloaded weldment profiles, the file location must be defined in the **Options** dialog.

Click **Options** , **System Options, File Locations**.

In **Show folders for:** select **Weldment Profiles** and click **Add**.

Browse to the Weldment Profiles folder and click **OK**.



Weldment Files

Besides the weldment profiles location, other external files referenced by SOLIDWORKS for weldments include:

- **Weldment Cut List Templates**
  (install_dir\lang\<language>\)

  Cut list template files (*.sldwldtbt) define the columns that appear in a cut list table.

- **Weldment Property File**
  (\\ProgramData\SOLIDWORKS\<SOLIDWORKS version>\lang\<language>\weldments)

  The weldmentproperties.txt file controls the list of property names that can be added from the drop down menu in the Cut List Properties dialog.

Inserting Structural Member

Inserting a structural member follows this general procedure:

1. Use a combination of 2D and 3D sketches to lay out the paths for the structural members.
2. Activate the **Structural Member** feature.
3. Specify a profile.
4. Select sketch segments to form a group (see *Groups* on page 19).
5. Specify the corner conditions between structural members, if necessary.
6. Specify the orientation and position of the profile if required.
7. Add additional groups if appropriate.

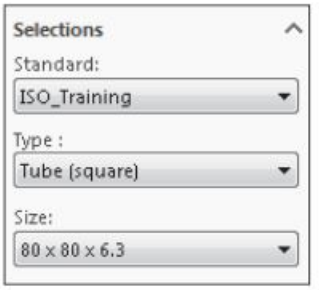
Where to Find It

- CommandManager: **Weldments > Structural Member**
- Menu: **Insert, Weldments, Structural Member**

---

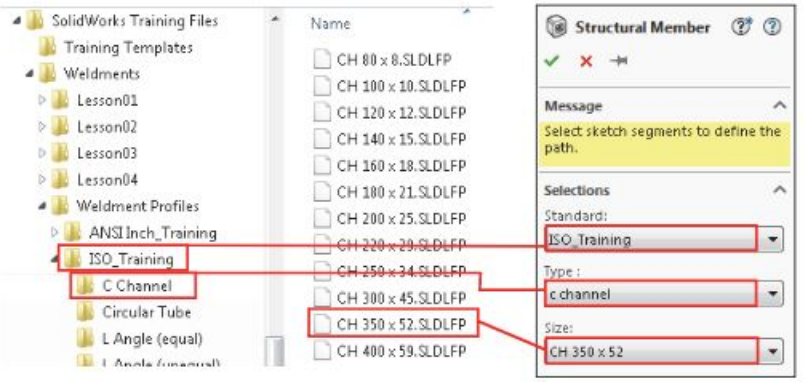
6 **Click Structural Member .**

7 **Specify a profile.**
Standard: ISO_Training
Type: Tube (square)
Size: 80 x 80 x 6.3



Weldment Profiles Folder Structure

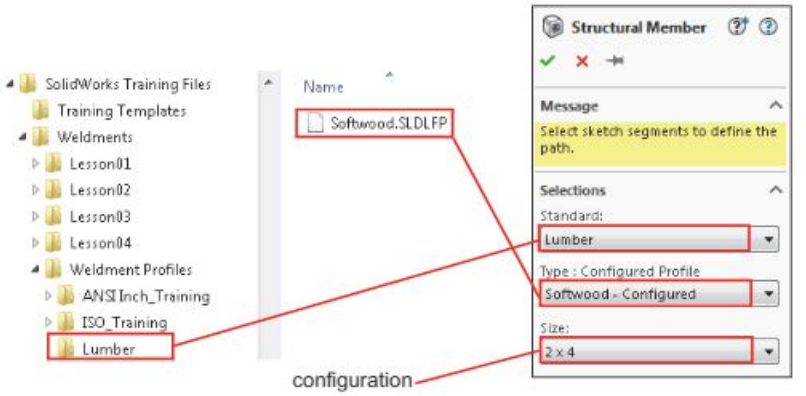
The structure of the weldment profiles folder corresponds to the selection fields in the Structural Member PropertyManager. An example of the typical file structure for a weldment profile (in this case for a CH 250 x 34 channel) is shown below.



Configured Profiles

For the default and downloaded profiles each file represents a single size of a specific type of structural member. In SOLIDWORKS 2014 and beyond, configured profiles can be used to represent multiple sizes of a profile within the same library feature part file.

If using a custom profile which utilizes configurations, the required folder structure is slightly different. No folder for "**Type**" is required; instead the name of the library feature part file should represent the "**Type**" of structural member and the configurations will populate the size field.



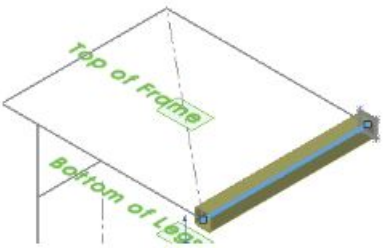
For more information on custom profiles, see *Custom Structural Member Profiles* on page 75.

---

8 Select the first path segment.

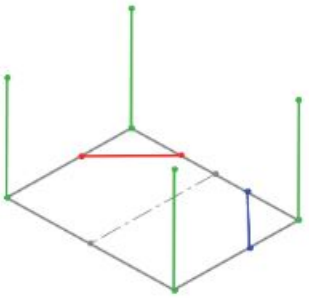
Select the first segment as shown.

The system creates a plane normal to the segment and applies profile sketch to it. A preview of the structural member appears.



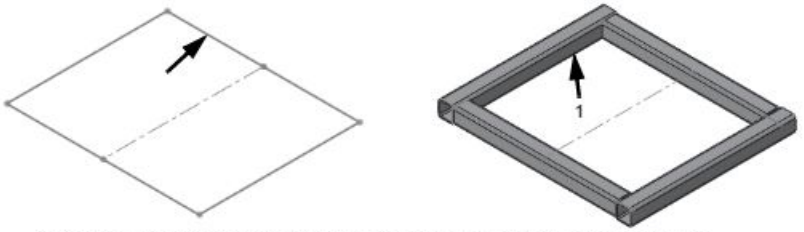
Groups

Sketch segments for a structural member feature are selected in **Groups.** Members of a group will share the same settings such as corner treatments, profile position, and profile orientation. In order for layout sketch segments to belong to the same group, they must either meet end-to-end, if corner treatments are required, or can be disconnected but parallel to each other.

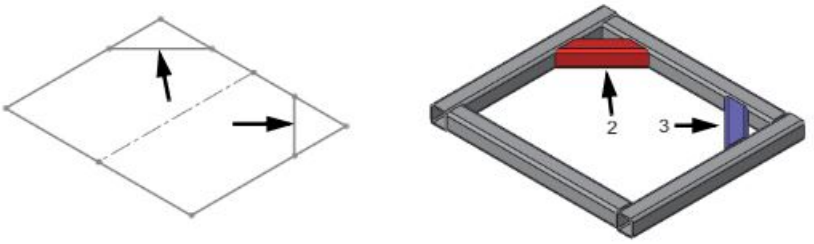


By using multiple groups within the same feature, the system is able to automatically perform trimming operations between the bodies being created. For more information see *Groups vs. Structural Members* on page 24.

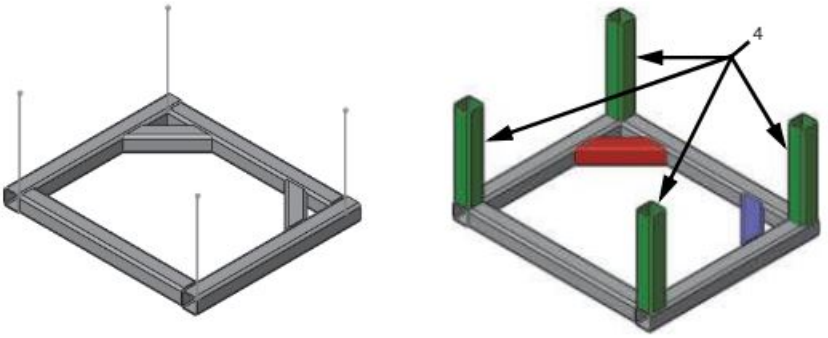
Connected; one group



Not connected (to each other) and not parallel; two more groups needed

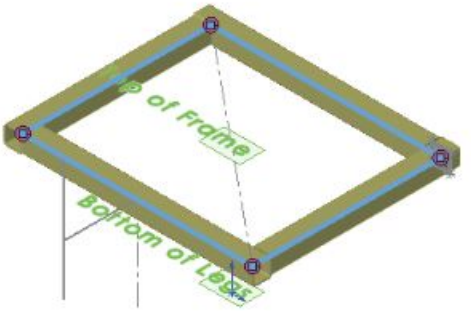


Disconnected (from each other) but parallel; a fourth group



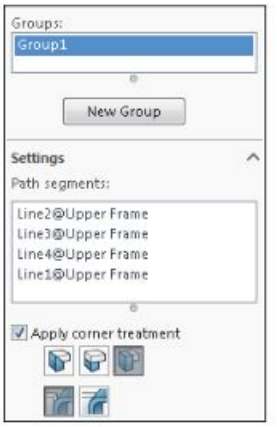
9 **Select segments for Group1.**
Select the remaining three segments that define the top of the frame.

This will be `Group1`.



10 **Corner treatment.**
Select **Apply corner treatment**.

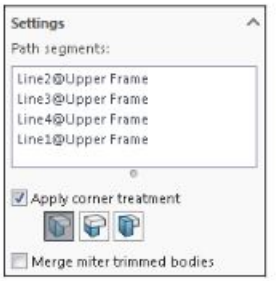
Click **End Butt2** and **Simple Cut** .



Corner Treatment Options

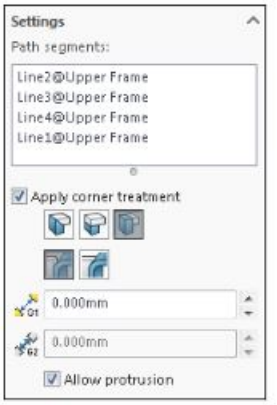
Corner treatment options will only be available when segments of a group meet at an endpoint. There will be slightly different settings available depending on the type of corner treatment selected.

- **End Miter**
  When selected, the option exists to **Merge miter trimmed bodies**. This would result in one body being created from the sketch segments.



- **End Butt1 & EndButt2**
  When selected, options for a **Simple** or **Coped** cut can be chosen.

  The option to **Allow protrusion** is visible. This allows a trimmed member to extend across the length of the sketch.



- **Arc Segments**
When arc segments are selected as part of a group, the option to merge arc segment bodies is visible. This allows for a single body to be created from multiple sketch segments such as would be desired to represent a bent tube.

- **Weld Gaps**
All corner treatment types include fields for adding weld gaps where the members meet. The first field is for gaps between the active group members while the second field can add weld gaps between the active group and another existing group.

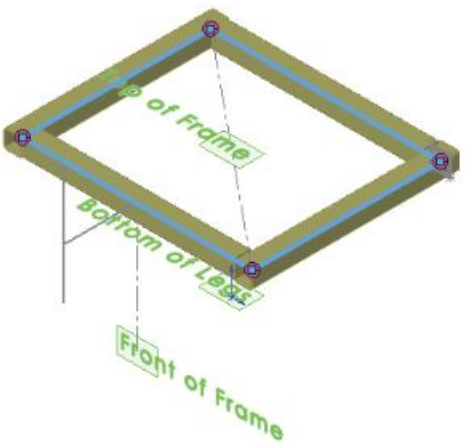
Individual Corner Treatments

The corner treatment options selected in the PropertyManager define the default condition, but individual corners can also be modified from the graphics area. To do this, click one of the spheres that appear at each corner and use the dialog to select the appropriate corner treatment and options.

This dialog also contains options to control the trim order when more than one group meet at the selected corner. For more information, see *Trim Order* on page 30.

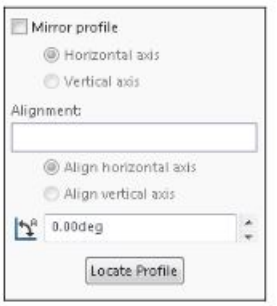
---

11 Corner treatment.
Change the corner treatments to match the illustration at right (the 2 segments parallel to the Front of Frame plane rest inside the other two).



Profile Position Settings

The lower section of the dialog contains some additional settings for positioning the profile:



Mirror Profile allows flipping of a profile. This is useful for non-symmetric profiles.

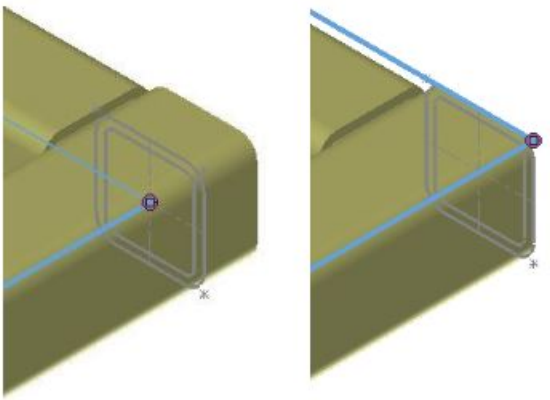
Alignment allows alignment of the profile to an edge or sketch segment, or at some angle from such an entity, or a specific angle from the current position can be entered.

Locate Profile lets you set the point on the profile where the layout sketch intersects, similar to a "pierce point" in sweeping. By default the Origin of the profile is pierced by the layout. Once the Locate Profile button is selected, any sketch point or vertex can be selected from the profile for alignment with the path segment.

---

12 Locating the profile.

Click **Locate Profile**. The display zooms in to show the profile sketch. Click the point at the virtual sharp of the upper-right corner. The sketch profile is repositioned.

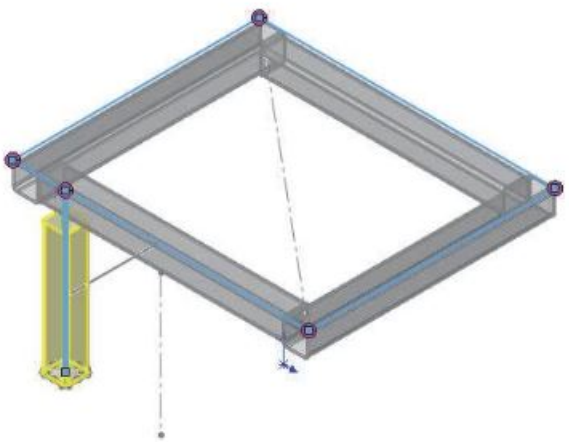


13 New group.

Click **New Group** to add a second set of components using the same profile.

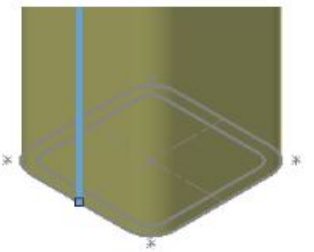
Select the vertical leg segment for Group2.

Notice the preview of the body shows it is being trimmed by Group1.



14 Locate Profile.

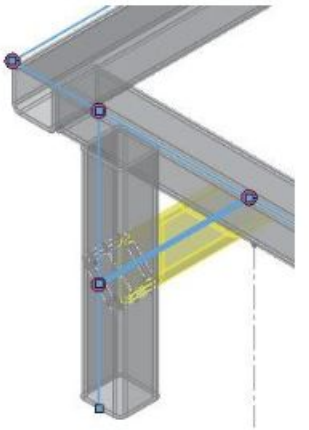
Locate the profile as shown.



15 Another group.

Create another group (Group3), for the angled leg support segment.

Locate the profile similarly.

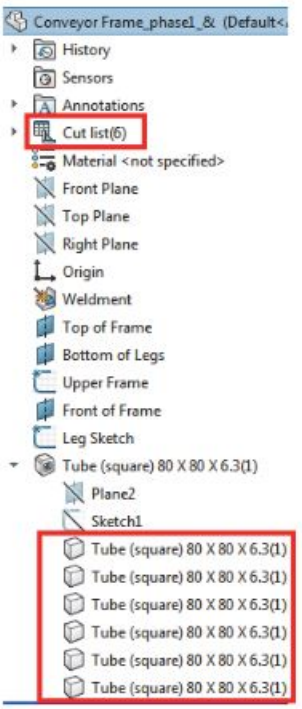


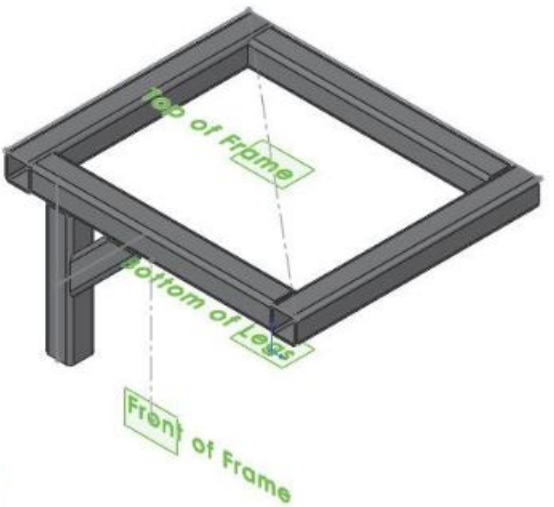
16 Results.

Click **OK** ✔ to finish the structural member.

Six separate solid bodies result from this feature.

Click **Save** .



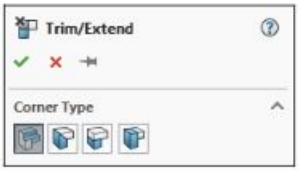


Groups vs. Structural Members

When the elements of a weldment structure use the same profile, they can exist within the same structural member feature, albeit in different groups, depending on the layout. When members are created within the same feature, trimming between segments is achieved automatically. It is best practice to take advantage of groups within the same feature whenever possible.

But what if there are different profiles required? Since only one profile can be used within a structural member feature, separate features must be created. When this is the case, manual trimming is often required between members.

Introducing: Trim/Extend



Trim/Extend is a specialized tool for manually trimming structural members and creating corner treatments. The same set of corner treatments that are available when inserting structural members are available within this command and can be used to create end miters or end butts between members of different sizes or types. In addition, an **End Trim** option will simply trim or extend a body to selected geometry.

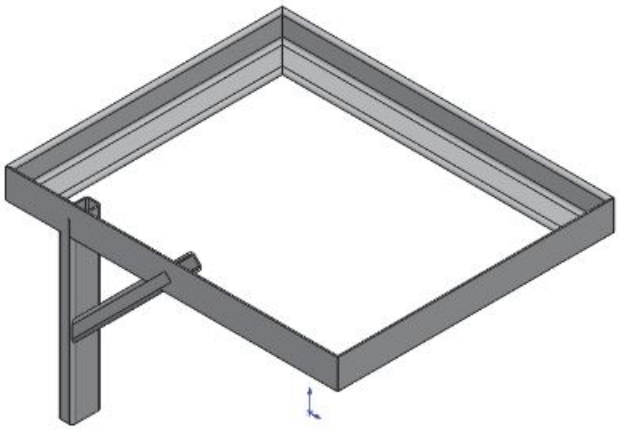
Where to Find It

- CommandManager: **Weldments > Trim/Extend**
- Menu: **Insert, Weldments, Trim/Extend**

---

1 Open a part.

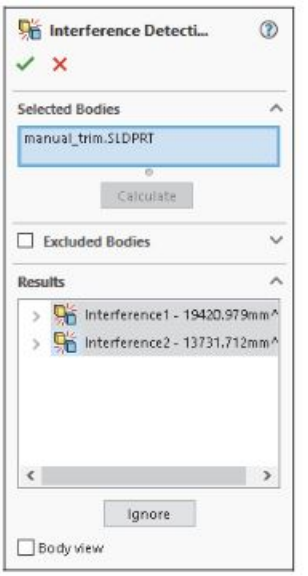
Open the file `manual_trim.sldprt`. It is similar to the `Conveyor Frame` part, but uses different profiles for the frame top and leg/brace combination. These bodies could *not* be created within the same structural member feature due to the different structural member profile.

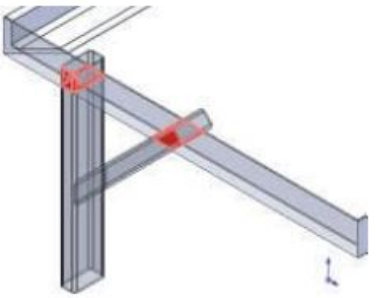


2 Check interference.

The interference detection tool works with assembly components *and* bodies. Click **Interference Detection** and view the results. Click **OK**.

Trimming will eliminate these interferences.



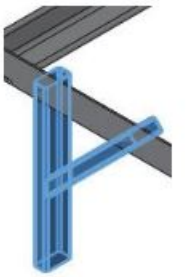


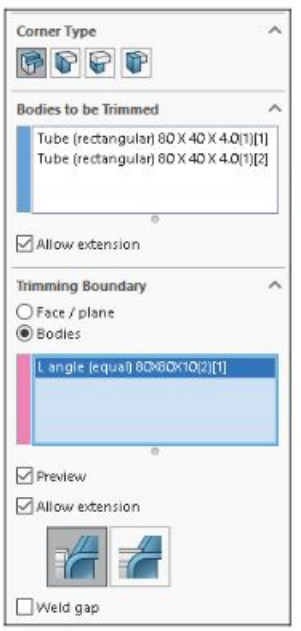
3 Trim.

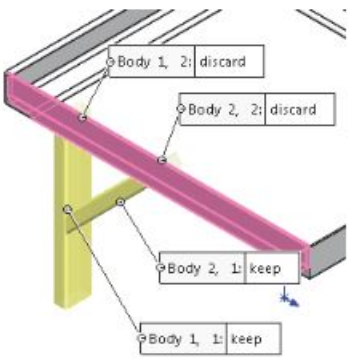
Click **Trim/Extend** .

Use the following settings:

- **End Trim** .
- **For Bodies to be Trimmed, select the leg and angled brace components.**
- **For Trimming Boundary, use Bodies, and pick the horizontal angle component.**







Trim/Extend Options

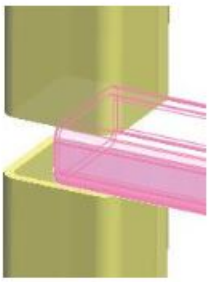
Trim/Extend options vary slightly depending on the **Corner Type** selected, similar to options in the **Structural Member** feature (see *Corner Treatment Options* on page 20). Some *unique* options in this command are listed below:

- **Allow extension**
  - For **Bodies to be Trimmed**, this option allows members to be extended as well as trimmed to match the Trimming Boundary (the Trimming Boundary is displayed in pink).

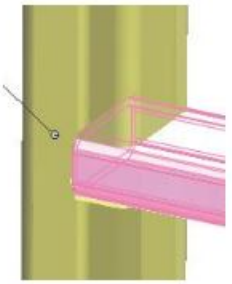
Bodies to be Trimmed, Allow extension selected

Bodies to be Trimmed, Allow extension cleared

  - For **Trimming Boundary**, **Allow Extension** will virtually extend trimming boundary selections to completely cut through members as shown below (the Trimming Boundary is displayed in pink).



Trimming Boundary, Allow extension selected



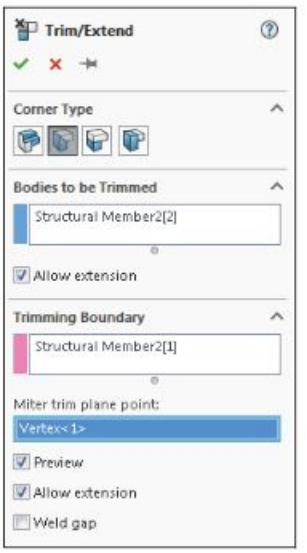
Trimming Boundary, Allow extension cleared

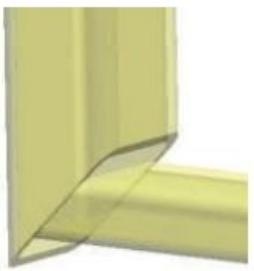
- **Face/plane or Bodies**
  When using the **End Trim** corner type, the **Trimming Boundary** can be individually selected faces and planes, or entire structural member bodies. If trimming to geometry that was not created with a structural member feature, faces must be selected for the trimming boundary.

Tip

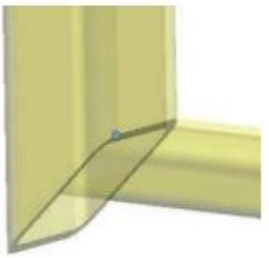
On complex weldment models, consider using **Face/Plane** selections for faster performance.

■ **Miter trim plane point**
When using the **End Miter** corner type, a vertex can be selected to define where the miter between members should start. When members of different sizes are coming to a mitered corner, often the default corner condition may not be desirable as shown below.





Default miter

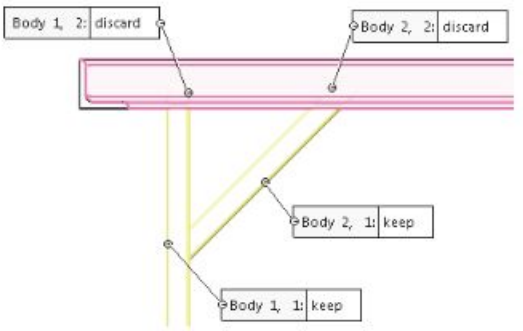


Miter with vertex selected

■ **Callouts**
Callouts appear in the graphics area when a trim boundary splits a member into multiple pieces. Clicking the callout toggles the option to **keep** or **discard** the new bodies.

---

4 Callouts.
Use the callouts to **keep** the 2 lower pieces and **discard** the small pieces that protrude through the top of the frame.

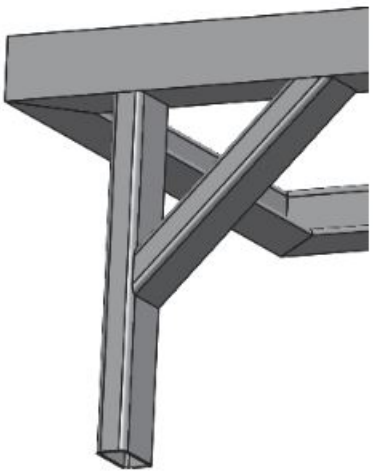


5 Complete the command.

Click **OK** ✔.

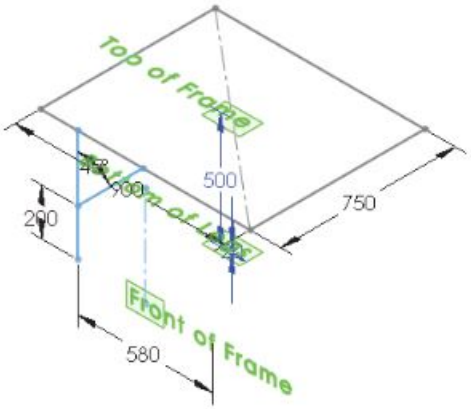
The result is seen at right.

Close the file without saving.



Sketch Considerations

Layout sketches are the foundation of working with structural members. You should keep in mind how the groups of structural members will be formed when constructing a weldment layout. Here are some things to consider when creating a layout sketch:



- Take advantage of patterns and symmetry.

  Mirroring or patterning bodies is an easy way to create similar pieces within the part while keeping sketch requirements at a minimum.

  For the `Conveyor Frame`, we are only constructing the leg and brace members for one corner and then will mirror the bodies to complete the frame.

- In order to be part of the same group, path segments must be either:

  • Connected end-to-end.

  • Disconnected but parallel to each other.

  Sketch segments do not have be in the same sketch to belong to the same group.

- You can use 2D, 3D, or a combination of both types of sketches.

  You have to balance the ease of sketching against any benefit that might come from having all the paths in one sketch. The conveyor frame we are working on has two 2D sketches: one for the top of the frame, and one for the leg. This could have been done in a single 3D sketch.

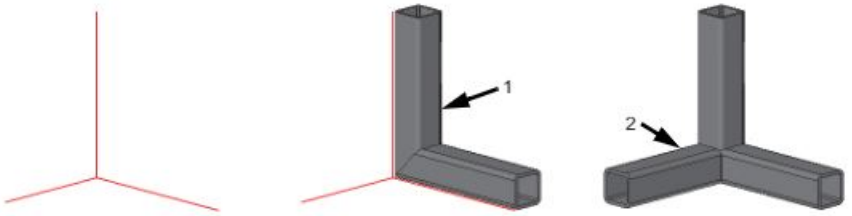
- Layout sketch segments can be re-used for multiple structural member features.

  Profile position settings can be used to modify how multiple members lie along a single sketch segment (see *Profile Position Settings* on page 22).

- The profile is positioned at the *start* point of the path segment selected first for a group.

  The start point is the first point placed when drawing the sketch segment. This has implications when working with asymmetrical profiles, although profile position options can be used to modify profile orientation as well.

- You cannot select more than two path segments that share a common vertex in a single operation. To create the corner shown below, you will need to use two groups.

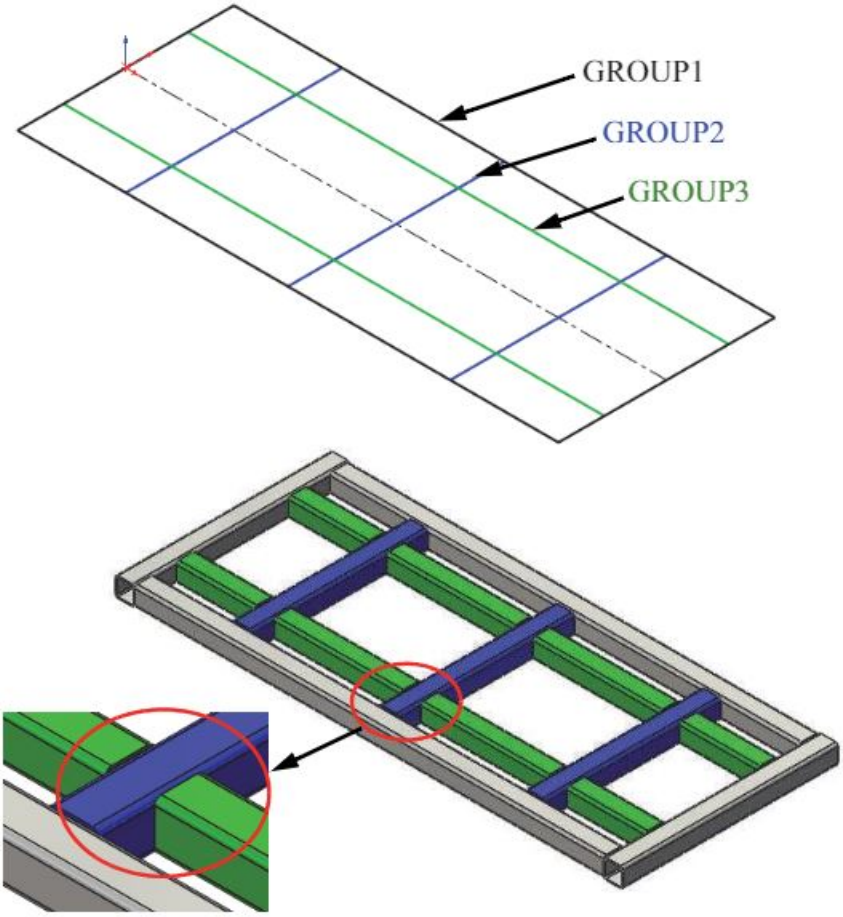


When multiple groups meet at a corner, **Trim Order** options can be adjusted to modify how the groups are trimmed.

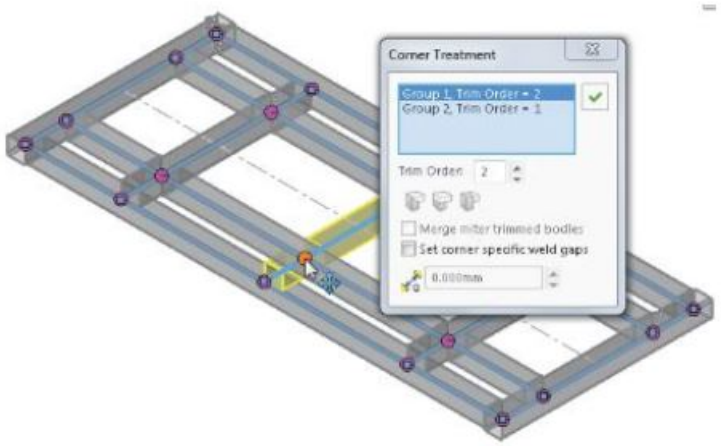
Trim Order

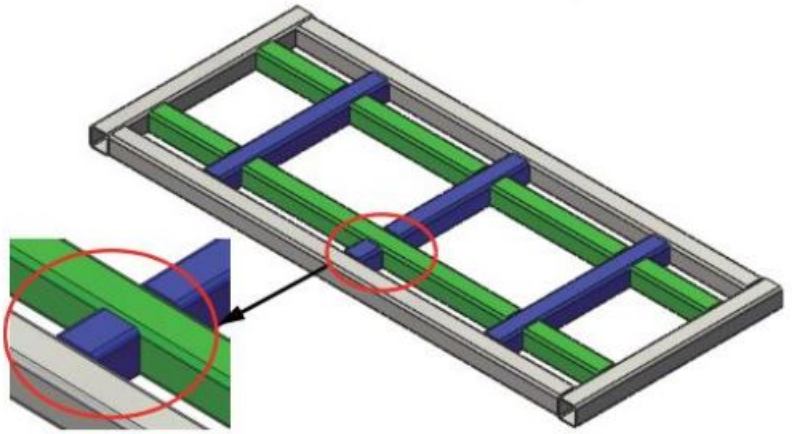
The order in which groups are added determines how the segments will be trimmed at their intersections. By default, the first selected group will be left full length and the following groups will be trimmed to meet it.

For example, in the illustration below, the three vertical blue lines and two horizontal green lines intersect in the middle of the structure. By selecting the blue lines into a group first, the two green lines are divided at the intersections with blue lines.



The **Trim Order** for individual corners can be modified using same dialog for modifying individual corner treatments (see *Individual Corner Treatments* on page 21). By switching the **Trim Order** for the first group at the intersection to be trimmed second and the second group to be trimmed first, the blue segment trims around the green at the selected corner.





The trim order can also be set to be the same for multiple groups at the corner. This would result in having all corner members trim to each other at the same time, such as the example below.

Adding Plates and Holes

Although structural members are the usually primary features of a weldment model, conventional feature types can also be used to create geometry in a weldment part. The `Conveyor Frame` will have base plates with holes welded to the bottom of each leg. To create this piece, we'll use an **Extruded Boss** and **Hole Wizard** feature.

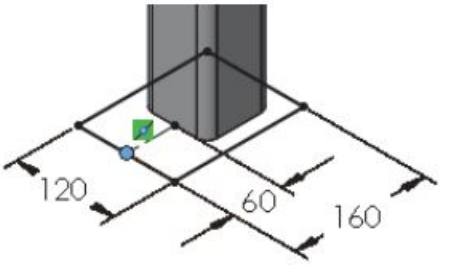
Common conventional features used with weldments are found on the **Weldments** toolbar for ease of access. These are the same commands that can be launched from the **Features** toolbar.

---

1 **Access the `Conveyor Frame`.**
This is the file we worked on previously.

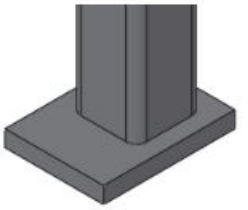
2 **Sketch the plate.**
Select the plane `Bottom of Legs` and open a **Sketch**.

Sketch a **Rectangle** as shown.



3 **Extrude.**
Click **Extruded Boss/Base**.

Use the end condition **Blind** and a **Depth** of **20mm**. Extrude downward, away from the legs.

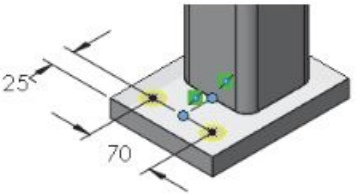


Note

Since this model is defined as a weldment, the **Merge Result** option automatically cleared to generate this feature as a separate solid body in the part (see *Weldment Feature* on page 9).

4 **Add the holes.**
Use the **Hole Wizard** to add two clearance holes for **M20** bolts.

Locate the holes as shown in the illustration.



Gussets and End Caps

Gussets and end caps are common features in weldments, but building them manually can tedious. The weldment environment has specialized tools that greatly simplify and speed up creating these features.

Introducing: Gusset

Gussets are plates added between existing members in a weldment. To insert a gusset two faces must be selected whose included angle is defined as: 0° < included angle < 180°.

Gusset Profile and Thickness

There are two gusset profiles: Polygonal and Triangular. Dimension values in the PropertyManager correspond to callouts displayed on the profile icons.

Polygonal

Triangular

In addition, you can add a **Chamfer**

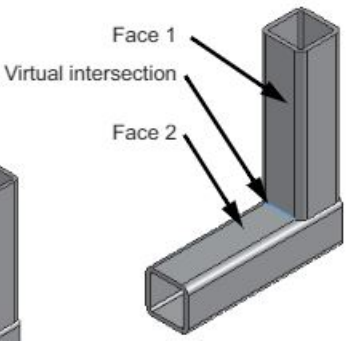
to the root corner of a gusset for weld bead clearance.

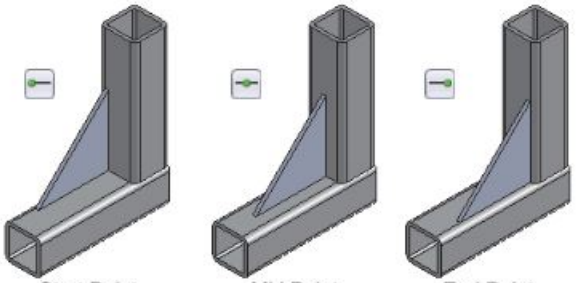
The thickness of the gusset is set in the same manner as for rib features. In the icons, the black line indicates the location selected for the gusset and the blue lines indicate how the thickness will be added from that location.

Inner Side	Both Sides	Outer Side

Locating the Gusset

When you select two faces, the system computes their virtual intersection. The gusset is located along that intersection. There are three possible positions:




Start Point Mid Point End Point

An offset distance can be specified from whatever position you select.

When applying a gusset to piping or tubing, the location will automatically be at the center.

Note

The use of gussets is not limited to weldment type parts. You can use the gusset feature in any part whether multibody or not. Gussets will always be created as a separate solid body.

Where to Find It

- CommandManager: **Weldments > Gusset**
- Menu: **Insert, Weldments, Gusset**

---

5 Insert a gusset.

Click **Gusset**.

Click **Polygonal Profile**.

Set **d1** and **d2** to **125mm**.

Set **d3** to **25mm**.

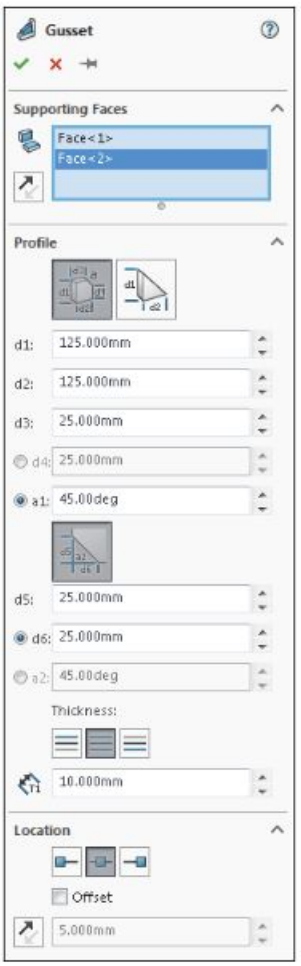
Set **Profile Angle** (**a1**) to **45°**.

Add an internal **Chamfer** with **d5**=**d6**=**25mm**.

Set **Gusset Thickness** to **10mm** and click **Both Sides**.

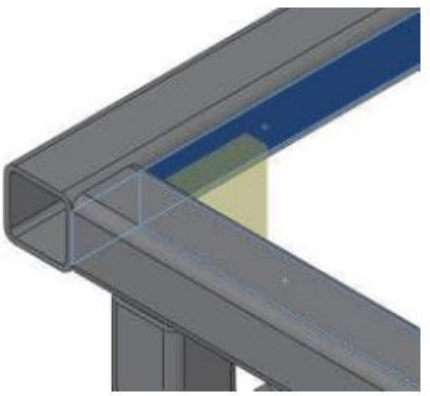
For **Location**, click **Mid Point**.

Select the faces shown in the illustration below.



Tip

If you use **Select Other**, be careful not to select the interior face of the tube.



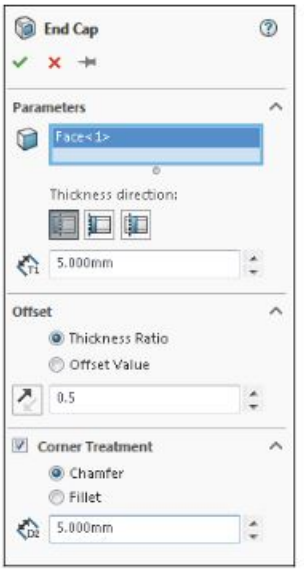
6 **Click OK** ✔.
The results are shown in the illustration at right.

Introducing: End Cap

End caps are metal plates that are welded across the openings of tubes and pipes. They are generally used to close off the openings in order to keep out dirt, debris, or other contaminants. To create an end cap feature, select the end face of a structural member you wish to cap.

End Cap Parameters

The size and shape of the end cap is primarily determined by the structural member face it's applied to. Several options are available within the End Cap PropertyManager to control specifics of how the end cap is created.



- **Thickness Direction**

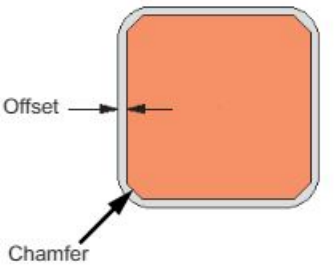
  Outward - applies end cap to end of the existing structural member.

  Inward - applies end cap thickness toward the existing member. The member is shortened to make room for the cap.

  Internal - adds the end cap to the inside of the member. An additional field becomes available with this option to define an offset from the end face of the member.

- **Offset**
  The profile of the end cap is determined by offsetting the faces of the structural member. The offset can be defined by a thickness ratio or a specific value. When creating an Outward or Inward cap, the offset is created from the outside faces of the tube or pipe, while with an Internal cap, the inside faces are used.

The default offset is one half of the wall thickness.



- **Corner Treatment**
  Use the corner treatment options to specify a chamfer or fillet at the corners of the end cap profile. Without corner treatments, a simple rectangular end cap is created for tube members.

Where to Find It

- CommandManager: **Weldments > End Cap**
- Menu: **Insert, Weldments, End Cap**

Note

Gussets and end caps are children of the structural members they reference. Deleting a structural member deletes any associated gussets or end caps.

---

7 Insert an end cap.

Click **End Cap** .

Click **Thickness direction**: **Outward**.

Note

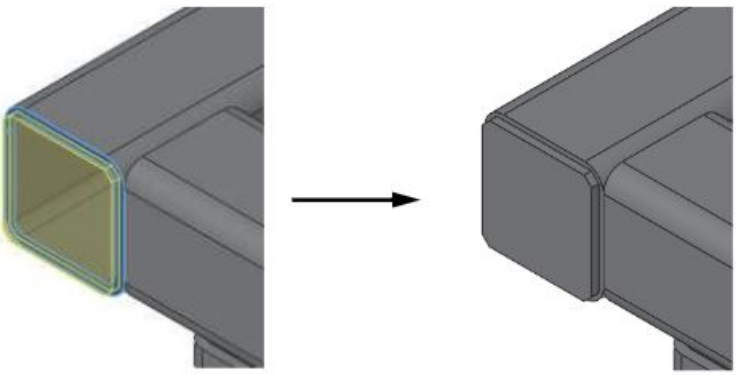
Using **Inward** here would keep the overall frame size the same by shortening the tube by the cap thickness.

Set **Thickness** to **5mm**.

Under **Offset**, click **Thickness Ratio** and verify it is set to **0.5**.

Click **Corner Treatment** and select **Chamfer.** Set the **Chamfer Distance** to **5mm**.

Select the end face of the tube as shown in the illustration below. Click **OK** .



Using Symmetry

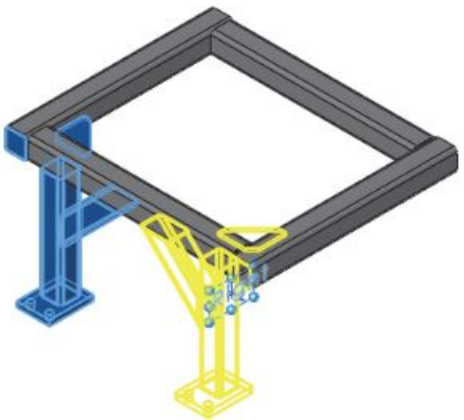
Mirror and pattern features can be used within a weldment model in the same way as they are used in conventional parts. The primary difference when using these features in a weldment is that you will most often be patterning individual **Bodies** rather than **Features**.

8 Mirror.

Click **Mirror**. For the **Mirror Face/Plane**, select the Right Plane.

Under **Bodies to Mirror**, select the leg, the angled brace, the plate, the gusset and the end cap.

Click **OK** ✔.



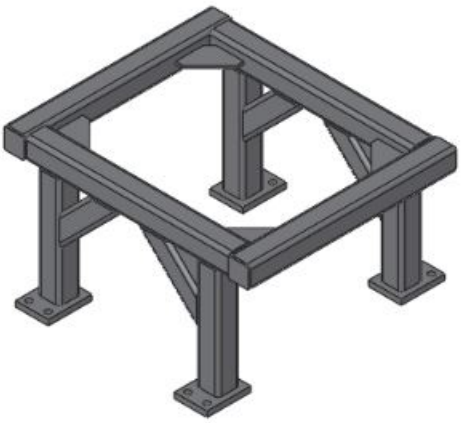
Note

Mirroring *features* instead of *bodies* for this operation would not work. There is only one structural member *feature*, but it created multiple *bodies*. We only want to pattern some of the bodies from that feature. Also, it is best practice to use the **Bodies to Mirror/Pattern** if it is indeed an entire solid body that is being duplicated. This will provide better performance and results.

9 Mirror again.

Use the Front Plane to mirror the front bodies as shown.

10 Save the part.



Advantages of a Multibody Part

Weldment structural members and features provide a quick and simple way to generate multiple pieces of a welded structure. Consider what would be required to build the Conveyor Frame as an assembly:

- If using bottom-up techniques, each unique piece would need to be created as a separate part, then inserted and mated into place. If changes were required once the assembly was complete, separate files would need to be modified and mates may need to be updated.
- If top-down assembly techniques were used, updates may automatically transfer to parts created in the context of the assembly, but the complex file relationships may be difficult to manage or result in slow performance.

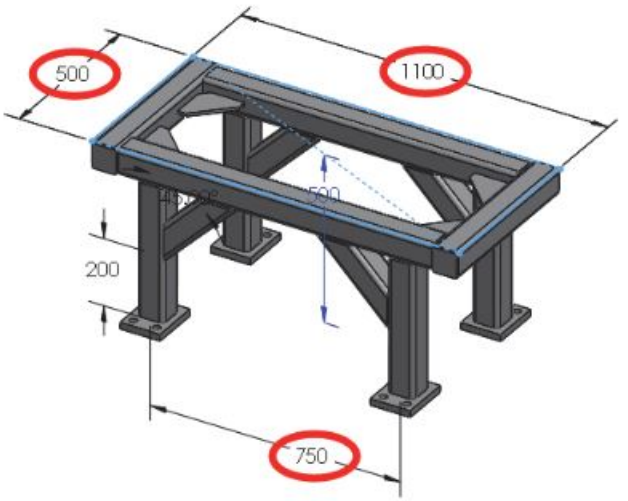
When working with a weldment as a multibody part, these limitations are eliminated. There is no need for multiple files or mates, and making changes to multiple pieces at once is as easy as modifying a layout sketch or feature.

---

11 Make changes to the Conveyor Frame.

Change the dimensions of the Upper Frame and Leg Sketch as shown.

Rebuild the part.

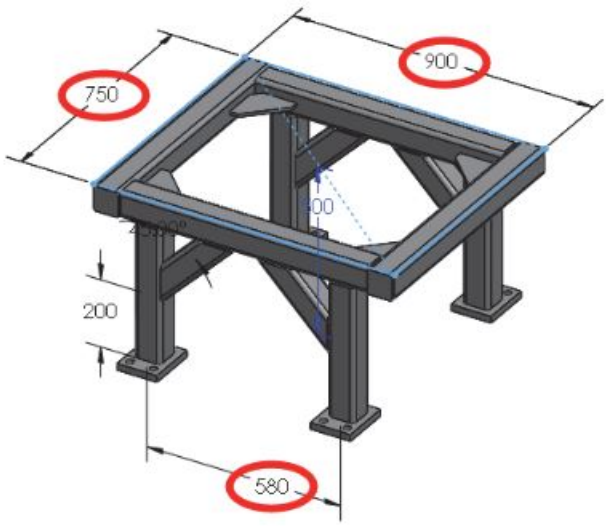


Members are resized and repositioned to match the layout. All corner conditions and profile locations are maintained.

12 Undo changes.
Change back to the original dimensions.

Tip

If the file was previously saved, you can simply **Reload** the last saved version from the **File** menu.



13 Save and close the part.

Tip

Because of the simplicity of making changes, weldments models are very easy to configure. For instance, if we produced several sizes of a similar `Conveyor Frame`, we could simply configure the dimensions or structural member profiles in the model to represent different frames.

Limitations of a Multibody Part

While there are some benefits to creating multiple pieces of a structure in a multibody part, there are also some limitations that can only be solved by working within an assembly. The primary limitations include:

- Cannot easily reposition components or simulate moving parts.

  Bodies within a part can be moved by modifying sketches and features or by making use of the **Move/Copy Bodies** feature, but dynamic movement in the graphics area is not possible.

- Slower performance than assemblies when dealing with large models.

  Assemblies have many options to aid in the performance of dealing with large file sets. Loading lightweight, using large assembly mode, and making use of simplified configurations are all efficient ways to increase performance for assemblies, but with part models the options are more limited. Utilizing the **Freeze Bar** and suppressing features may aid in part performance, but feature order and parent child relationships must be considered. See the *SOLIDWORKS Assembly Modeling* course for more information on lightweight and performance options with assemblies.

- Detail drawings of weldment bodies will be tied to the weldment part file.

  Bodies within a weldment *can* be detailed individually, but the model referenced will be the weldment part. Company standards must be considered to determine if this is appropriate. For more information on detailing weldment bodies, see *Drawing Views of Individual Bodies* on page 108.

Exercise 1: Sign Holder

Create the Sign Holder frame pictured here by using SOLIDWORKS weldment features. A part is provided with the layout sketches, or you can choose to start from scratch.

This lab reinforces the following skills:

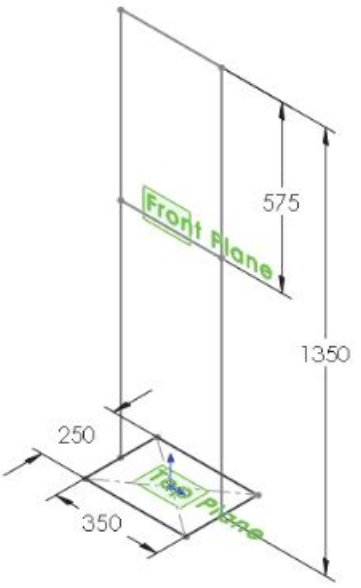
- *Weldment Configuration Options* on page 10
- *Inserting Structural Member* on page 17
- *Groups* on page 19
- *Corner Treatment Options* on page 20
- *Profile Position Settings* on page 22
- *Gusset* on page 33
- *Using Symmetry* on page 37

Units: **MMGS**

Procedure

1 Create a new part.
To begin this model from scratch, create a new part using millimeters as the units.

2 Layout sketches.
Use the default Top and Front planes to create the layout sketches as shown.



Note

To make use of existing layout sketches, open the part Sign Holder from the Lesson01\Exercises folder.

3 Modify configurations options.
This simple frame will not require any machining after welding, so there is no need for the automated derived configurations.

In **Options** ⚙, **Document Properties**, select **Weldments.**

Clear the checkbox for **Create derived configurations**.

Note

Since this setting is a **Document Property** this change only affects the current document. Document property settings can be saved in document templates to establish default settings.

4 Download profiles.

Follow the instructions on page 15 - page 16 to download the ISO standard profiles. Be sure to **Rename** the folder as stated and define the **File Location** in **Options** .

5 Add a structural member feature.

Click **Structural Member** .

Use the following selections for the structural member profile:

Standard: ISO_Training

Type: Tube (square)

Size: 20 x 20 x 2.0

6 Group1.

Select the rectangular line segments that lie on the Top Plane to be Group1.

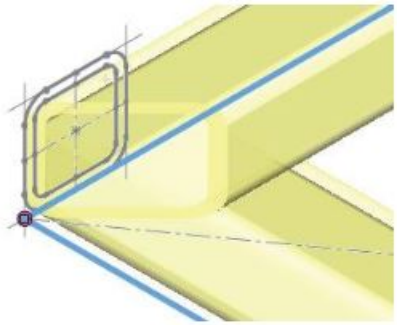
Note

These segments meet end-to-end and require the same settings, such as corner treatments between them and profile location, so they should be created as a single group.

7 Group1 settings.

Apply a corner treatment between the members and select **End Miter** as the type.

Click the **Locate Profile** button at the bottom of the PropertyManager. Locate the profile for this group so that the members sit above and inside the layout dimensions.



Note

The position of the profile in the model is determined by the first sketch segment picked. The profile may be in a different position than pictured depending on how you made your selections.

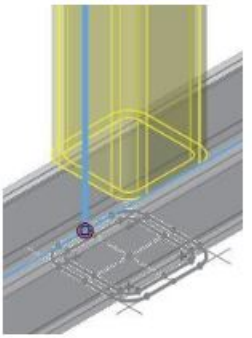
8 Group2.

Click the **New Group** button.

Select the 3 lines that make up the outside of the frame as Group2.

9 Group2 settings.

Select an **End Miter** corner treatment and locate the profile so the members sit inside the layout dimensions.



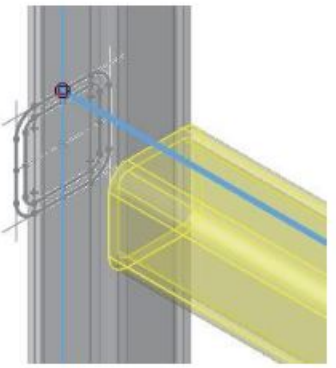
10 Group3.

Click **New Group**.

Select the last horizontal line as Group3.

Locate the profile to sit below the layout line.

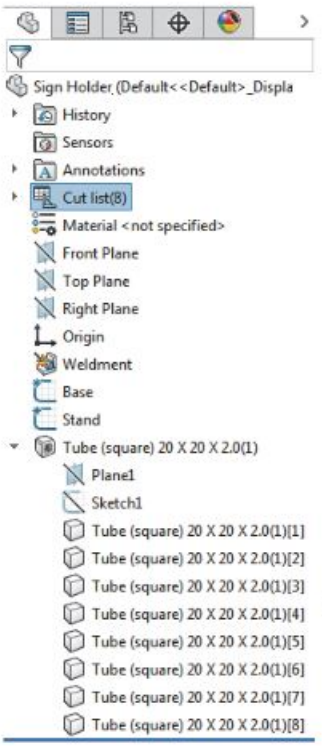
Click **OK** ✔.



11 Results.

The **Weldment** feature has automatically been added to the FeatureManager tree.

Eight (8) separate bodies are created in the part.



12 Add a cut feature.

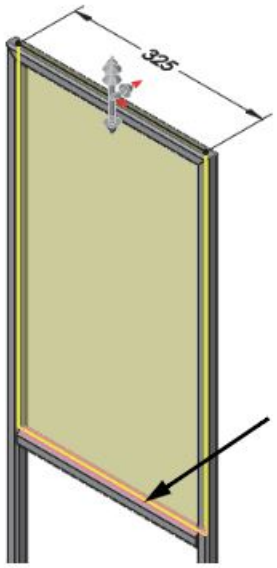
Hide the layout sketches.

Sketch a **Midpoint Line** on the top face of the part, centered on the Origin, **325mm** long.

Click **Extruded Cut** .

Select **Thin Feature**, **Midplane**, **Thickness** = **2.5mm.**

For **Direction1**, select an end condition of **Up to Surface**. Select the top face of the member shown as the face to cut to.



13 Add a gusset.

Click **Gusset** .

Select the face of the vertical frame as shown first, then the top of the center horizontal member.

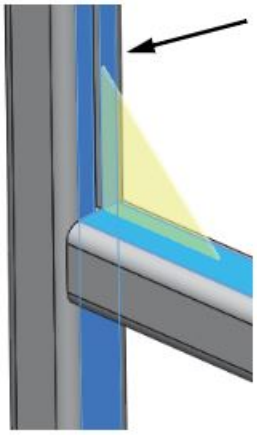
Note

Selecting the face on the back side of the cut first helps define the gusset location.

The profile is **Triangular**, **35mm** x **35mm**.

The thickness is **1mm**, applied to the **Outer Side** and the location is at the **End Point** .

Click **OK** .

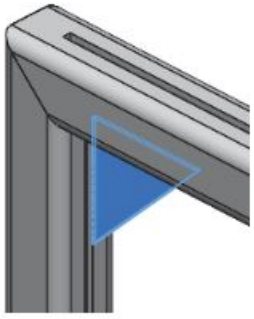


14 Add a second gusset.

Add a second gusset with the same dimensions and similar location in the upper corner of the frame.

15 Mirror gussets.

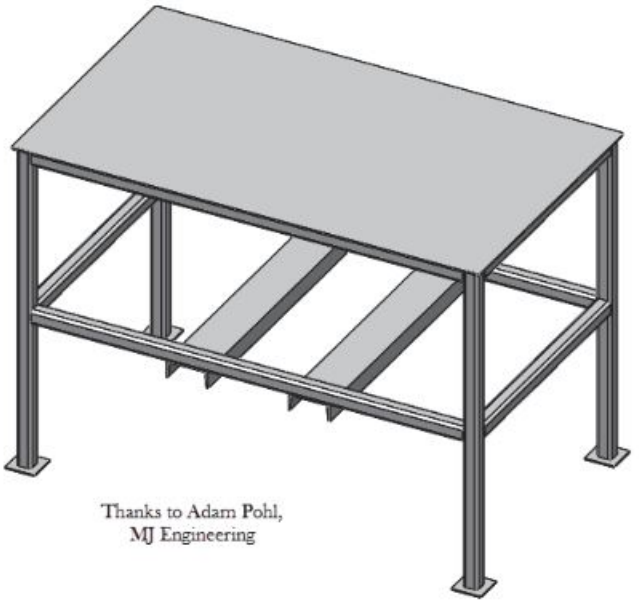
Mirror the gusset bodies over the Right Plane to add copies to the other side.



16 Save and close the part.

Exercise 2: Weld Table

Use structural members and plates to create the Weld Table shown, then change the dimensions to modify the table size.



This lab reinforces the following skills:

- *Inserting Structural Member* on page 17
- *Groups* on page 19
- *Profile Position Settings* on page 22
- *Trim Order* on page 30
- *Adding Plates and Holes* on page 32
- *Using Symmetry* on page 37

Units: **MMGS**

Procedure

1 Open an existing part.

Open the part Weld Table from the Lesson01\Exercises folder.

This part contains a single 3D sketch that will be used to define the table frame.

2 Add frame structural members.

Click **Structural Member** .

Use the following selections for the structural member profile:

Standard: ISO_Training

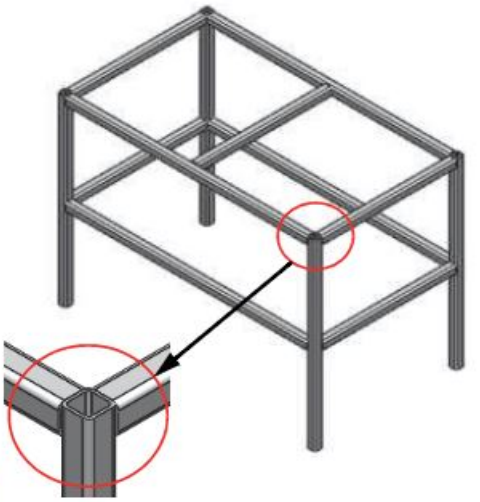
Type: Tube (square)

Size: 50 x 50 x 5.0

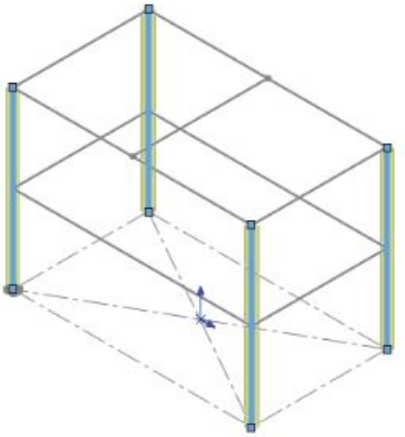
3 Group1.

All frame members will use the same profile, so they can be created within the same structural member feature. The sketch segments need to be selected in groups which will best determine the proper trim order and settings.

The legs of this frame run from top to bottom and the other members fit in between. Therefore, the legs will be the first group selected so all other members will trim to them.



Pick the 4 vertical leg segments to be `Group1`.

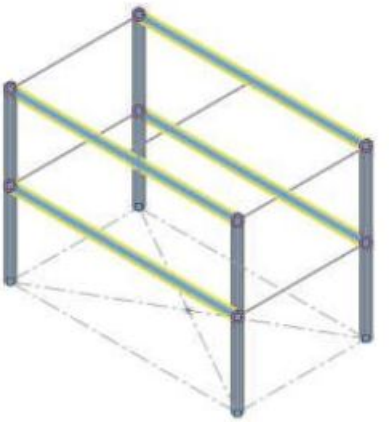


These segments are disconnected but parallel and require the same settings, such as profile location, so they can be created as a single group.

4 Group2.

Click **New Group**.

Pick the 4 long horizontal line segments for `Group2`.

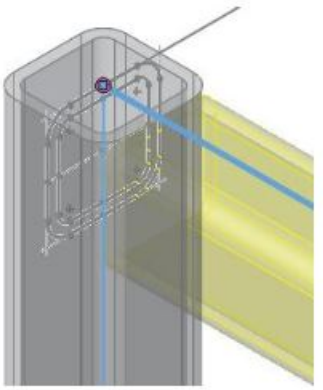


Note

Since the line segments for the top of the frame meet end-to-end and would share the same profile location, they would qualify for a group. However, since no corner treatment is needed, selecting the front and side members separately will allow for proper trimming to `Group1`.

5 Locate profile.

Locate the profile as shown. All members in the group will share this location.

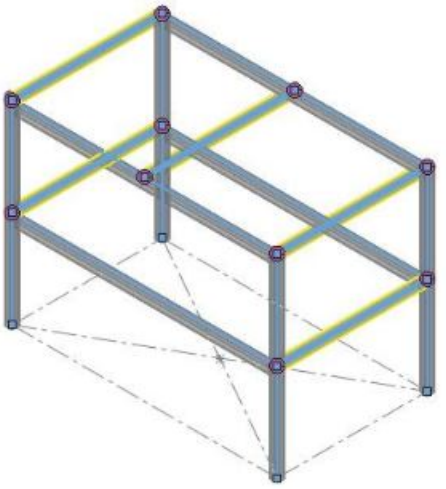


Note

The position of the profile in the model is determined by the first sketch segment picked.

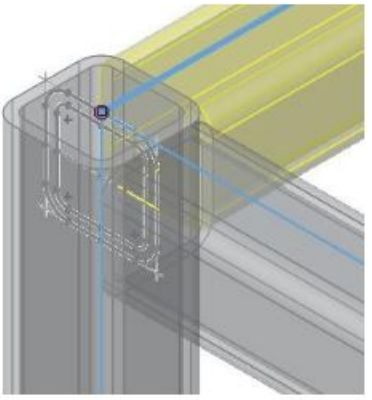
6 Group3.

Click **New Group**.



Pick the remaining sketch lines to form a group from the parallel segments.

Locate the profile as shown.



7 Results.

Click **OK** ✔ to complete the feature.

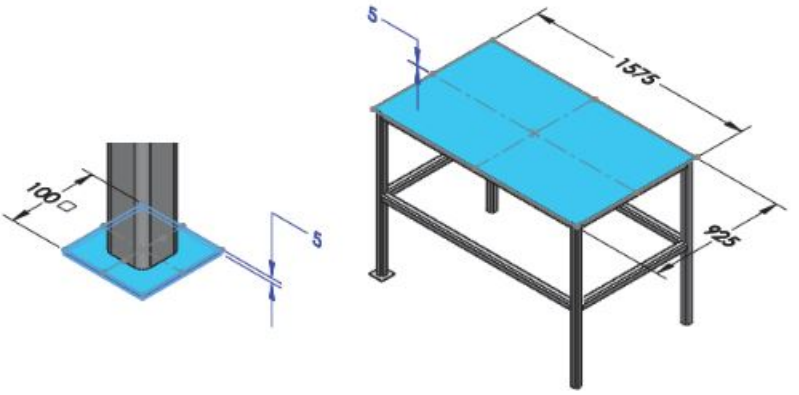
Thirteen (13) solid bodies are created in the model.

Hide the Frame Layout sketch.

8 Add plates.

Sketch and extrude the plates for the table as shown.

Use existing faces in the model as sketch planes.



9 Mirror bodies.

Mirror the foot plate across the Front and Right planes.

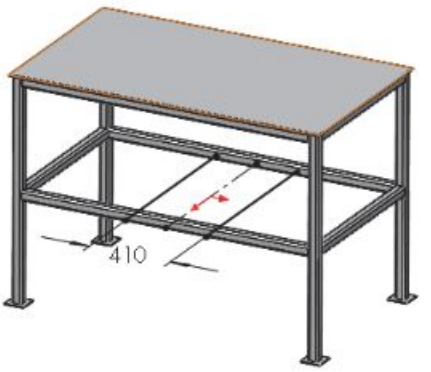
10 Add fillets.

Add **10mm** fillets to the corners of the top plate.

11 Channel layout sketch.

Create a new sketch on the bottom face of a cross brace member.

Sketch symmetrical line segments, **410mm** apart, as shown.



12 Channel structural members.

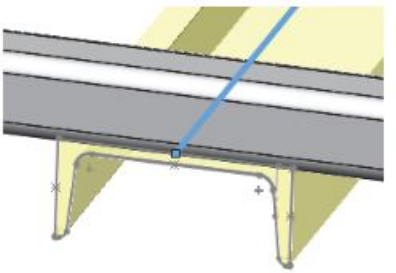
Add structural members to the layout using the following profile:

Standard: ISO_Training

Type: c channel

Size: CH 140 x 15

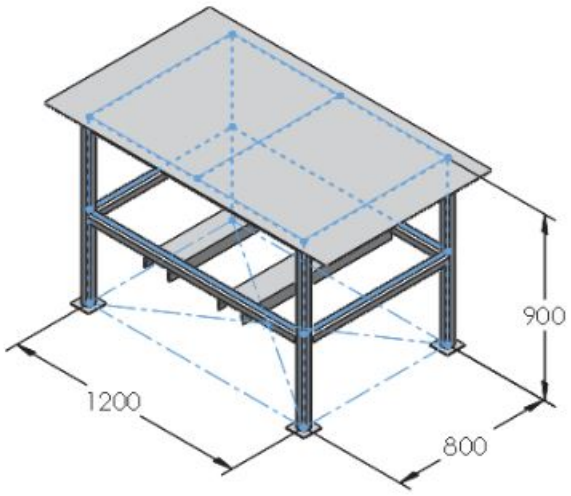
Position the profile as shown using settings.



13 Make a change.
The Weld Table is made from 20 separate pieces.

Modify the Frame Layout to the dimensions shown and **Rebuild** .

All frame structural members will update.



14 Save and close the part.

Exercise 3: Suspension Frame

Use SOLIDWORKS weldment features to create the Suspension Frame shown. Some structural member will require manual trimming.

This lab reinforces the following skills:

- *Inserting Structural Member* on page 17
- *Groups* on page 19
- *Profile Position Settings* on page 22
- *Trim/Extend* on page 24
- *Adding Plates and Holes* on page 32
- *Using Symmetry* on page 37

Units: **MMGS**

Procedure

1 Open part.

Open the existing part Suspension Frame from the Lesson01\Exercises folder.

2 Add frame structural members.

Click **Structural Member** .

Use the following selections for the structural member profile:

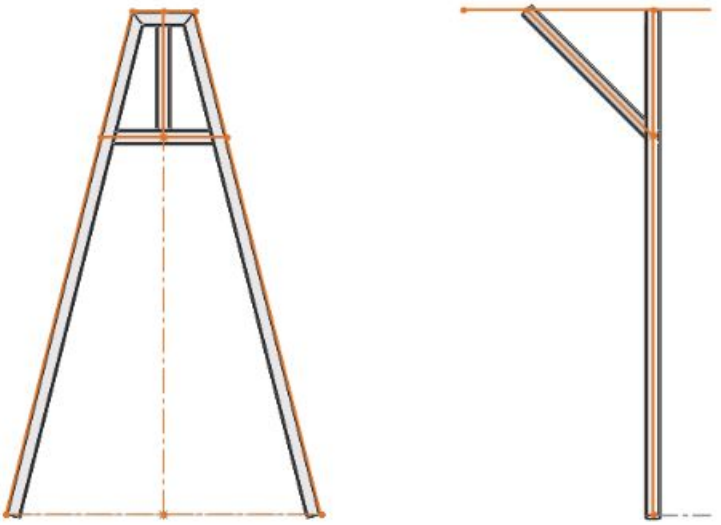
Standard: ISO_Training

Type: Tube (square)

Size: 70 x 70 x 4.0

3 Create groups.

The pieces of the side frame and angled brace share this profile, so they can be created within the same feature. Use groups to create the members as shown. The layout sketches are shown in color for clarity.



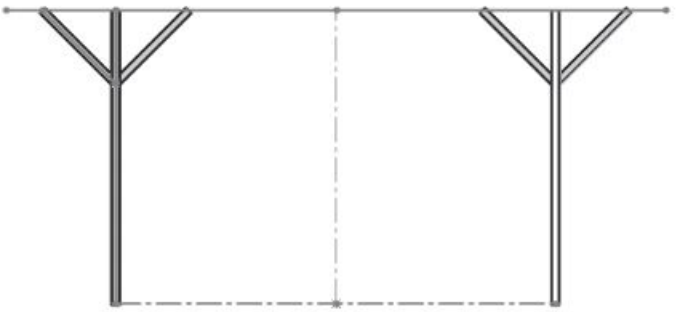
- Use settings to properly position the profile for each group as necessary.
- The order the groups are created will determine the trim order.
- Four groups will be required.
- This feature should create 6 separate solid bodies in the part.

Tip

To make changes to a group that already exists, simply select it in the **Groups** selection box to access its settings.

4 Mirror.

Use **Mirror** features to copy the brace and side frame bodies.



5 Add the top bar.

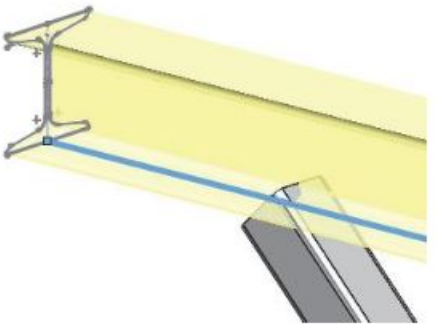
The top bar uses the following structural member profile:

Standard: ISO_Training

Type: SC Beam

Size: SC 120

Use settings to position the profile as shown.



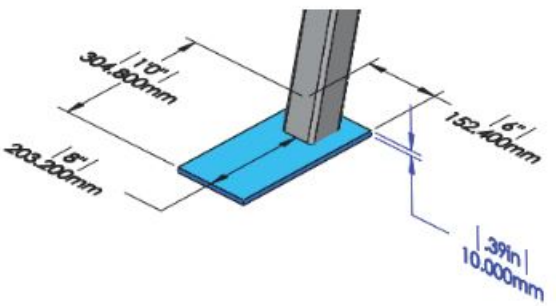
6 Trim/Extend.

Use the **Trim/Extend** tool to trim and extend the angled brace bodies to the top bar.

7 Foot plate profile.

Use the Top Plane to sketch the foot plate profile as shown.

Extrude the foot plate profile **10mm**. The direction should be away from the frame members.



Tip

Feet&Inches are shown as dual units for this part. If you prefer to enter dimension values in something other than the primary units, you can type abbreviations directly into the Modify dimension dialog or use the menu for **Units**. Abbreviations for feet include "**ft**" or an apostrophe ('). For inches, "**in**" can be used or a quotation mark (").

8 Mirror.

Use **Mirror** features to copy the foot plate body to the other corners of the frame.

9 Trim/Extend.

Trim the frame members to the top face of the foot plate.

Note

Faces, rather than bodies, must be selected when trimming to geometry that was not created with a structural member feature. **Face/Plane** selections are virtually extended to cut through all selected bodies to be trimmed.

10 Save and close the part.

Exercise 4: Evaporator Support

Create this evaporator support weldment by following the steps given.

This lab reinforces the following skills:

- *Inserting Structural Member* on page 17
- *Profile Position Settings* on page 22
- *Adding Plates and Holes* on page 32
- *Using Symmetry* on page 37

Units: **MMGS**

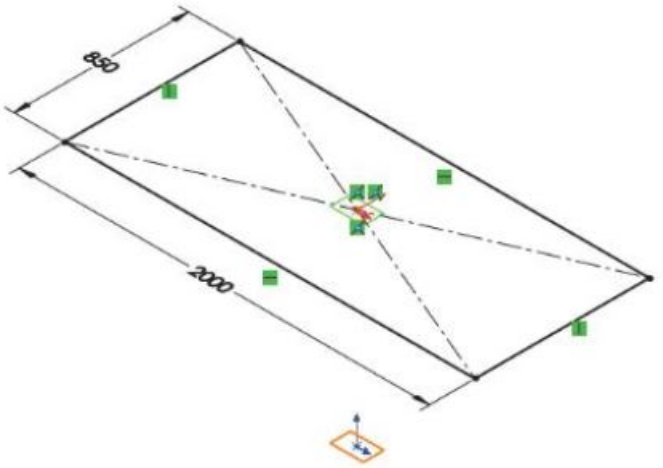
Procedure

1 Create a new part.
Open a new part using the Part_MM template.

2 Create new plane.
Create a **Plane** offset **1000mm** above the Top Plane.

3 Layout sketch for top of frame.

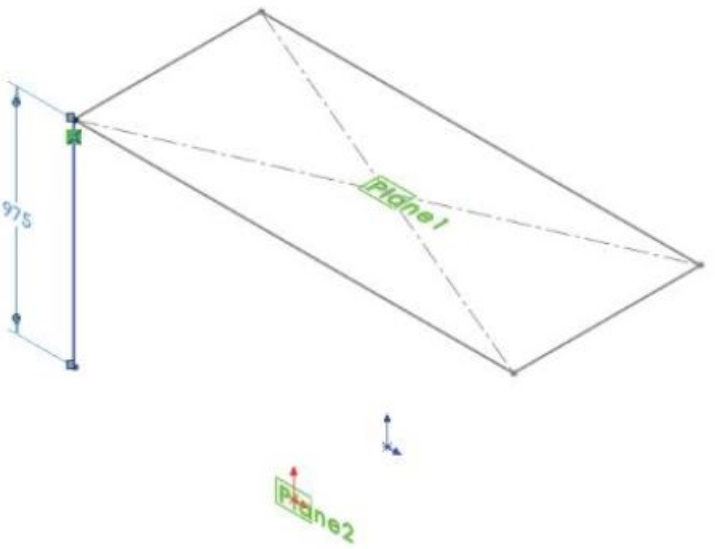
Sketching on the new plane, create a rectangle (2000 x 850) for the top of the frame as shown and exit the sketch.



4 Create sketch for leg.

Create another **Plane** that is **Parallel** to the Front Plane and **Coincident** to the front, left corner of the rectangle.

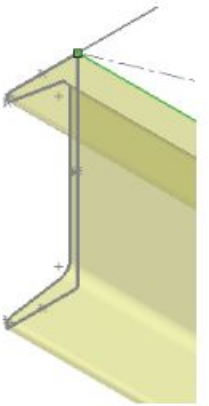
Sketch a line **975mm** long for the leg as shown.



5 Structural member - C Channel.

Use ISO_Training, c channel, CH 120 x 12 and insert a structural member at the top of the frame.

Use settings to locate the profile as shown.

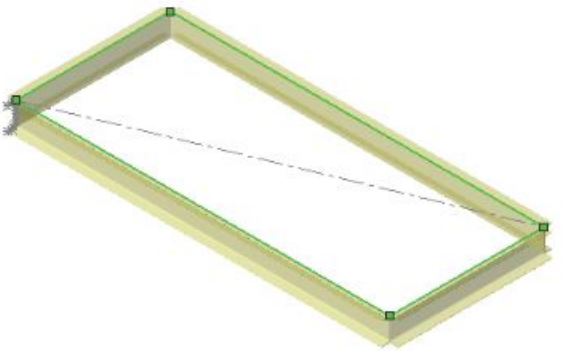


Note

The initial profile position may be different than pictured depending on how the layout sketch was created.

6 Complete the top of the frame.

Select remaining sketch segments.



Clear the **Apply corner treatment** check box.

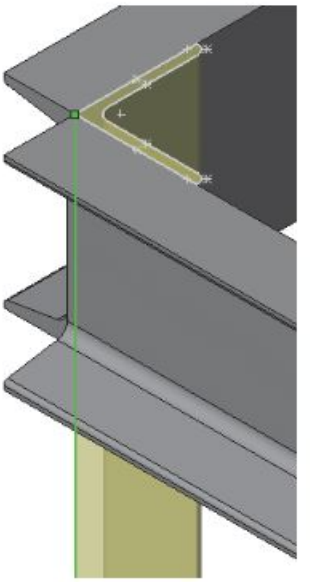
Note

When no corner treatment is used, the structural members simply follow the length of the path segments.

Click **OK** ✔.

7 Insert the structural member for the leg.

Use the L Angle (equal), 80 x 80 x 6 profile.



Locate the profile as shown.

Click **OK** ✔.

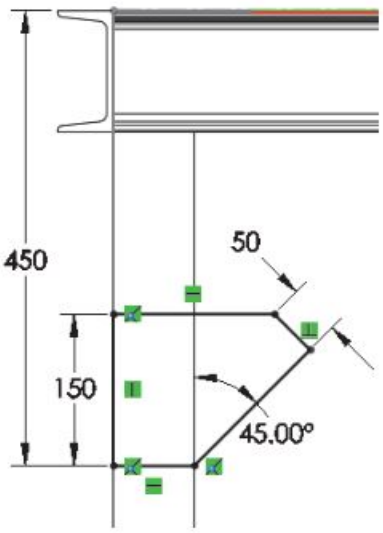
Note

The sketch is shown in color for clarity.

8 Make the reinforcing plate.

A gusset feature will not work here for two reasons:

- A gusset is created at the virtual intersection between 2 faces. This plate is not positioned at an intersection.
- The shape of the profile doesn't match either of the two that are available with the gusset feature.

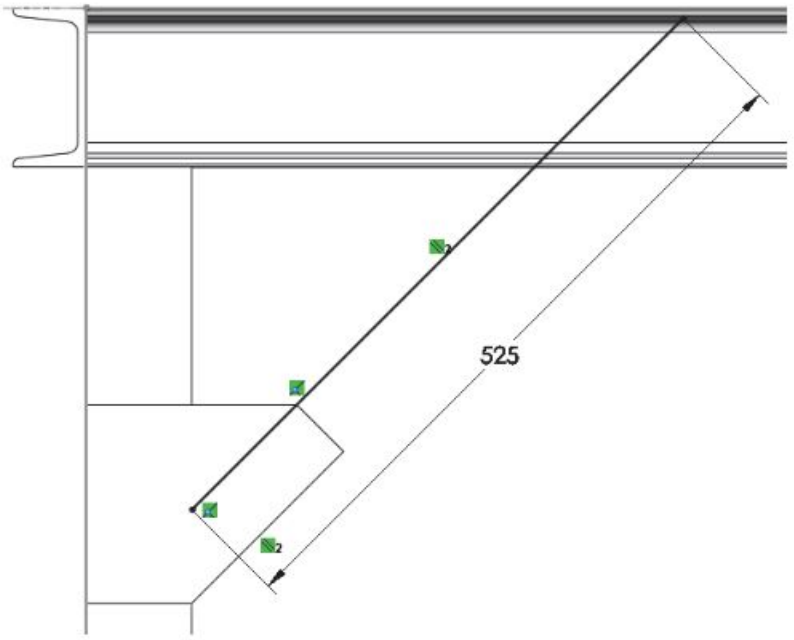


Create the sketch as shown on the outer face of the leg member.

Extrude it to a depth of **6mm**.

9 Sketch for the angled brace.

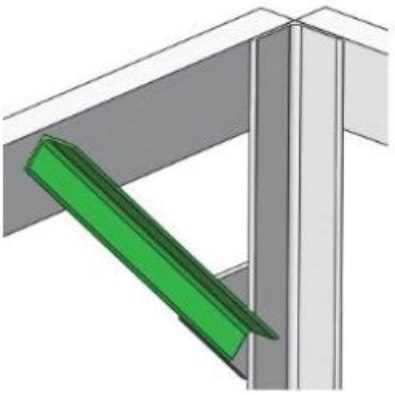
Open a sketch on the *rear* (inside) face of the reinforcing plate. Sketch a line as shown below. Note the various geometric relations necessary to fully define the sketch.

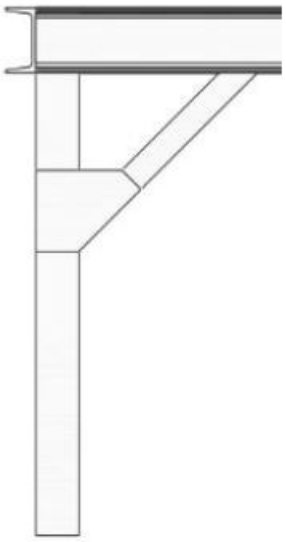


10 Structural member.

Use L Angle (equal), 50 x 50 x 4.

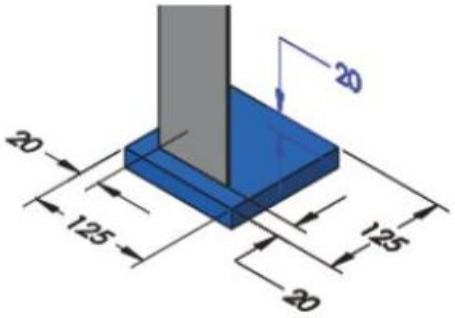
Note the orientation of the profile as shown in the view below.





11 Foot pad.

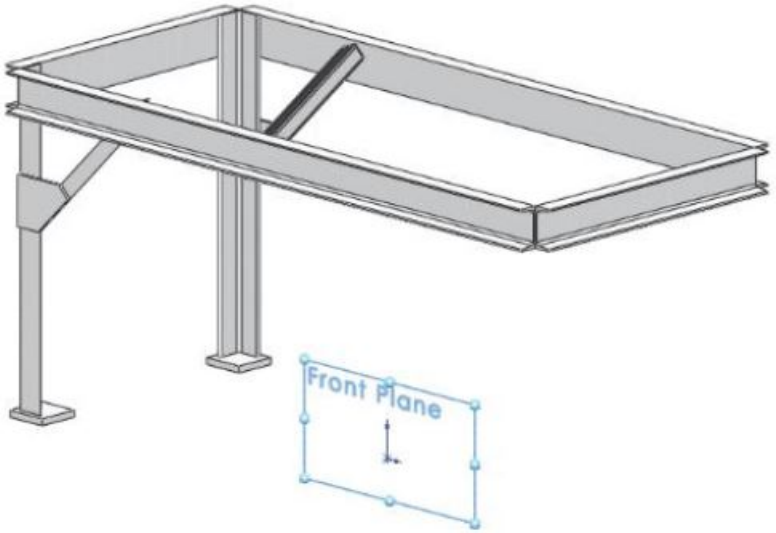
Create a foot pad using the dimensions shown at the right.



12 Mirror.

Mirror the leg bodies over the Front Plane.

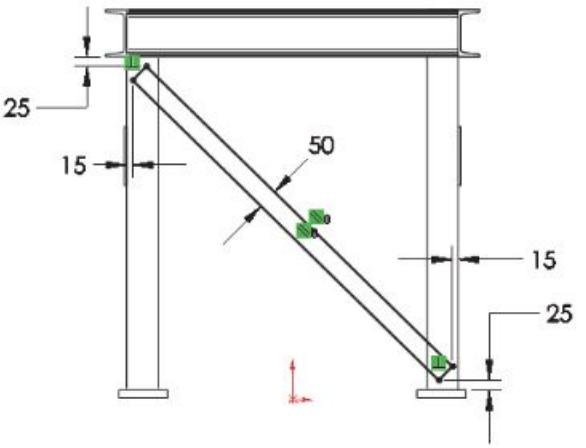
The results are shown in the illustration below.



13 Cross brace.

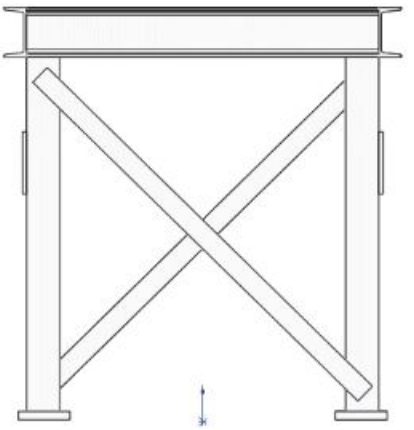
Sketch a **3 Point Corner Rectangle** and dimension it as shown.

Extrude it to a depth of **6mm**.



14 Repeat.

Make a second cross brace on the inside face of the legs.

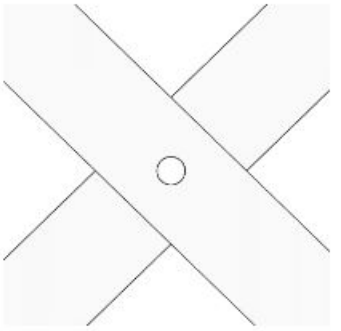


15 Hole Wizard.

Use the **Hole Wizard** to make a clearance hole for a **12mm** bolt through the two cross braces.

Tip

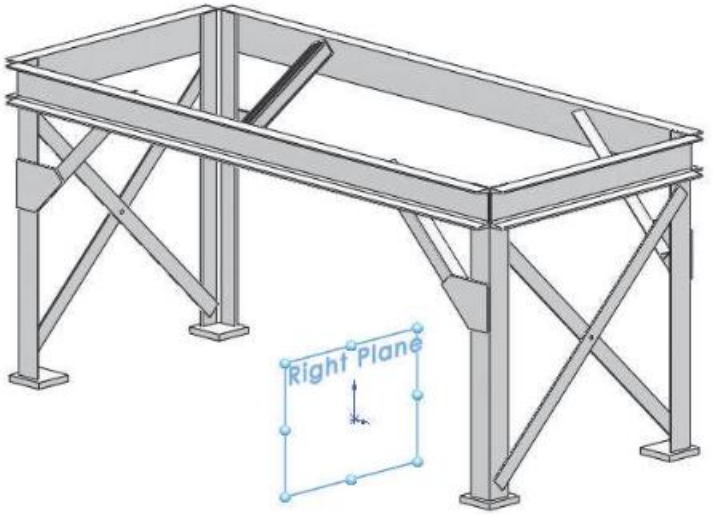
Consider using construction geometry in the hole wizard location sketch to center the hole.



16 Mirror.

Mirror both sets of leg components over the Right Plane.

The results are shown below.



17 Save and close all files.

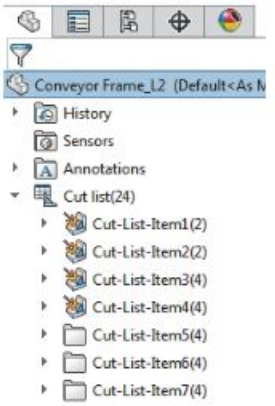
Lesson 2
Working with Weldments

Upon successful completion of this lesson, you will be able to:

- Manage the weldment cut list items and properties.
- Add cut list item properties using the Cut-List Properties dialog, the Weldment feature, and Bounding Boxes.
- Manually manage cut list items.
- Create and manage sub-weldments.
- Modify and create structural member profiles.
- Apply material to individual solid bodies.
- Insert existing parts into a weldment model.

Managing the Cut List

A cut list is like a bill of materials for a multibody part. The Cut list folder in the FeatureManager design tree is used to organize the bodies of a model for a cut list table. This is done by grouping like items together into *cut list item* sub folders. Each cut list item folder represents one line item in the cut list table. Custom properties can then be applied to the cut list items to communicate information in the table.



The number in parentheses following the cut list item name indicates how many bodies are grouped into that cut list item. The cut list item folder icons indicate how the bodies for that item were created:

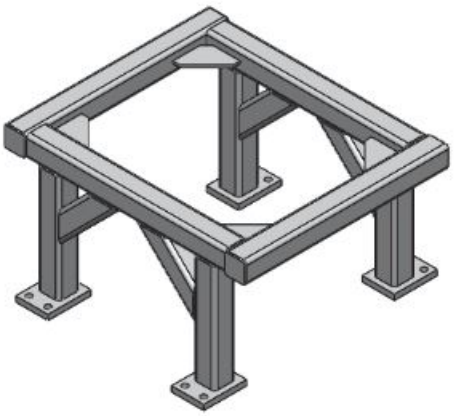
	Structural Member Bodies
	Sheet Metal Bodies
	Standard Feature Bodies

Note

Cut lists are automatically used when working in either a weldment or a sheet metal model. For specifics of managing sheet metal cut lists, see the *SOLIDWORKS Sheet Metal* course.

1 **Open the Conveyor Frame.**
Continue working with the model created in Lesson 1, or open the model provided in the Lesson02\Case Study folder, Conveyor Frame_L2.

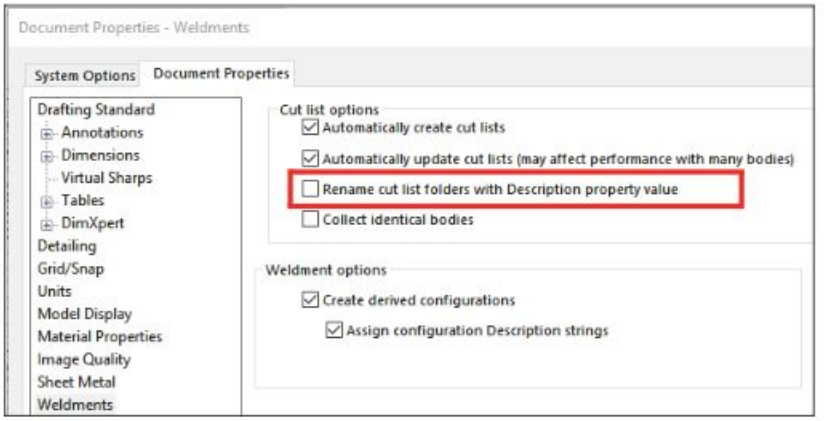
2 **Expand the Cut list folder.**
The 24 bodies of the Conveyor Frame have been grouped into 7 separate cut list items.



Cut List Item Names

Cut list items are given simple sequential names by default, but these can be renamed to assist in organizing the model. Cut list item names can be displayed in the cut list table if desired. Renaming cut list items can be done manually, or the Description property for each cut list item can be displayed as the folder name in the FeatureManager design tree.

The option to use the Description property as the cut list item name is located in **Document Properties**. If this option is selected, the cut list folders cannot be manually renamed.



Note

Changing **Document Properties** only affects the current document. Document property settings can be saved in document templates to establish default settings.

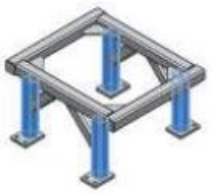
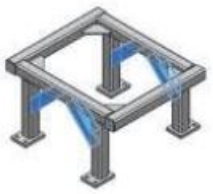
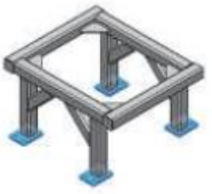
---

3 Renaming the cut list items.

The default cut list item names are not very descriptive. Use the following table as a guide for renaming the cut list item folders.

Use standard renaming techniques: a slow double-click on the text or highlight and press **F2**.

Tip Selecting a cut list item folder in the dialog or in the FeatureManager design tree will highlight the associated bodies in the model.

Name of Cut List Item	Selected Bodies	Name of Cut List Item	Selected Bodies
SIDE TUBES		FRONT-REAR TUBES	
LEGS		ANGLED BRACES	
BASE PLATES		GUSSETS	
END CAPS			

Accessing Properties

Properties for cut list items are accessed by right-clicking a cut list item folder and choosing **Properties** from the menu. For cut list items that are the result of a structural member feature or sheet metal feature, many properties will be automatically generated.

4 Access the cut list item properties.
Right-click a cut list item folder and click **Properties**.

Cut-List Properties Dialog

The Cut-List Properties dialog contains 3 tabs for viewing and modifying cut list item properties.

- **Cut List Summary**
  Access each cut list item in the left pane to view associated properties.
- **Properties Summary**
  Access each existing property name in the left pane to view how it's defined in each cut list item.
- **Cut List Table**
  Preview how the cut list table will appear when created. The **Table Template** field can be used to load different cut list table templates to preview.

Tip

Cut list items are listed in the order the bodies were created. Cut list items can be reordered by dragging the cut list item folders in the dialog or the FeatureManager design tree. The line items in the cut list table will follow this order. Cut lists can also be configured.

---

5 **Cut List Summary.**
All items have a MATERIAL and QUANTITY property.

The items containing bodies from the structural member feature have several additional properties.

Structural Member Properties

Cut list items that result from a structural member feature automatically capture additional information that would be common to display in a cut list table and used for producing the part. The additional properties created for structural member bodies are as shown:

LENGTH	Length of a single body
ANGLE1	Mitered angle cut at one end
ANGLE2	Mitered angle cut at other end
TOTAL LENGTH	Total length of all bodies using this profile
Description	Inherited from the profile library part

Adding Cut List Properties

There are several ways to add properties to cut list items:

- Use the Cut-List Properties dialog to add properties to individual items (see step **7** on page 68).
- Add properties to the Weldment feature to apply a property to all cut list items (see step **8** on page 69).
- Create a **Bounding Box** to automate size related properties for non-structural member bodies (see *Bounding Boxes in Weldments* on page 69).

Note

The cut list folder name can be linked to a cut list property as well, so that it can be included in BOMs, notes and sheet formats.

Tip

Another option is to use a Property Tab Builder Weldments template can be used to enter properties using a predefined form. See Help documentation on *Property Tab Builder* for more information.

---

6 Select the END CAPS cut list item.

Use the left pane in the Cut List Properties dialog to select the END CAPS cut list item.

7 Add a PARTNUMBER property.

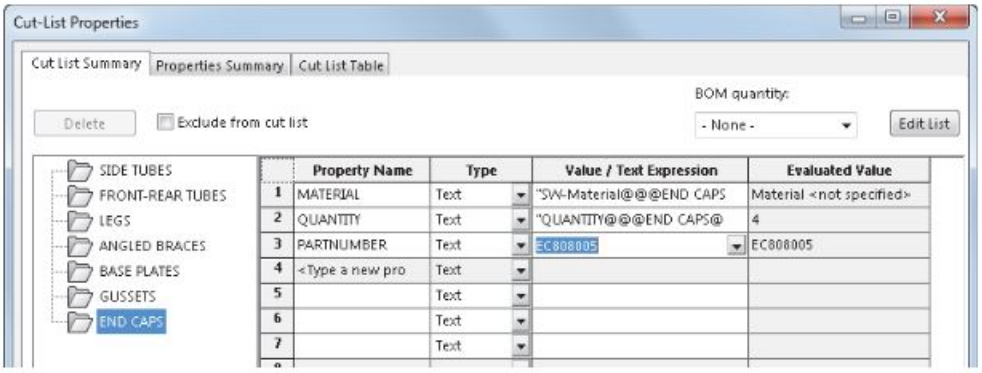
Click in the next available Property Name cell.

Use the menu to choose **PARTNUMBER** as the property name.

Click in the Value/Text Expression cell.

Type EC808005.

Click **OK** to close the dialog.



Tip

The list of properties associated with weldments is maintained in the weldmentproperties.txt file. It is located in \\ProgramData\SOLIDWORKS\<SOLIDWORKS version>\lang\english\weldments.

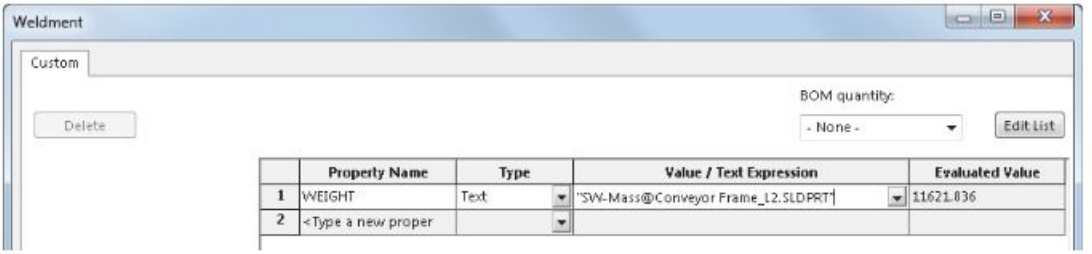
8 Add a WEIGHT property to *all* cut list items.

Right-click the **Weldment** feature and click **Properties**.

Use the menu in the **Property Name** cell to add a **WEIGHT** property.

Use the menu under **Value/Text Expression** to link to **Mass**.

Click **OK** to close the dialog. Click **Rebuild**.



9 Review the Cut List Properties.

Access the **Cut-List Properties** dialog and click the **Properties Summary** tab.

Select the **WEIGHT** property that now exists and verify each cut list item has a unique value for its mass.

Click **OK** to close the dialog.

Bounding Boxes in Weldments

Bounding boxes in weldments automate a 3D sketch surrounding a cut list item body. The 3D sketch represents the smallest box the body can fit into. Information regarding the size of the bounding box is automatically translated into the cut list item's properties.

The bounding box sketch is stored in the cut list item folder, where it can be shown and hidden.

A bounding box can be created for any cut list item, independent of the type of solid bodies in the cut list item.

Where to Find It

- FeatureManager design tree: Right-click a cut list item folder, click **Create Bounding Box**

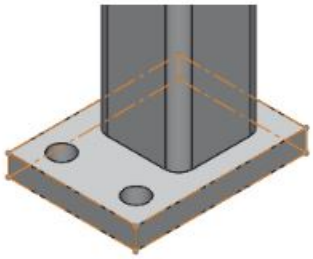
Editing a Bounding Box

If a bounding box exists for a cut list item, the options to **Edit Bounding Box** and **Delete Bounding Box** are available. Edit a bounding box to modify the default reference plane. This will adjust the direction of captured properties such as "Thickness".

Where to Find It

- FeatureManager design tree: Right-click a cut list item folder, click **Edit Bounding Box**

10 Create a Bounding Box for the Base Plate cut list item.



Right-click the BASE PLATES cut list item.

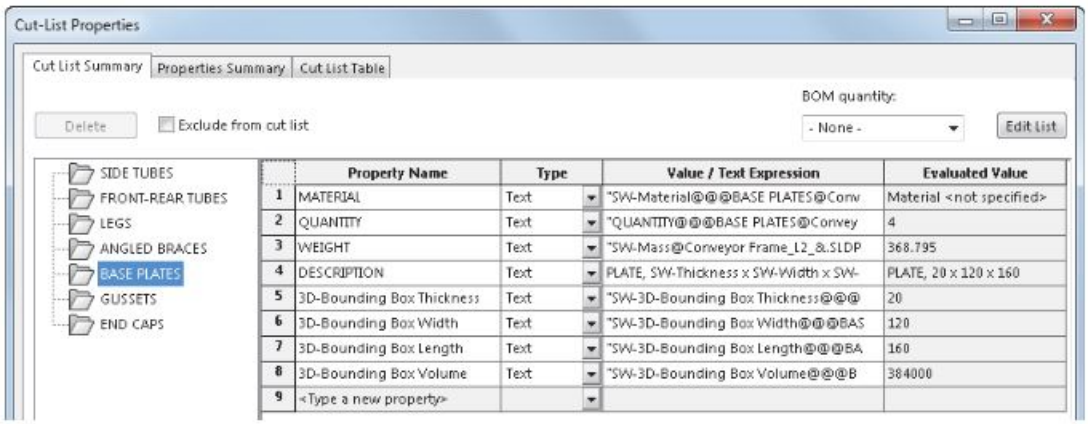
Click **Create Bounding Box**.

Expand the cut list item folder to view the 3D sketch that was created.

11 Review the Cut List Properties.

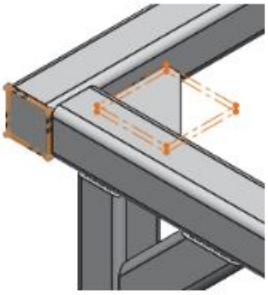
Access the **Cut List Properties** dialog.

Several cut list properties were created relating to the bounding box size including the thickness, width, length, and volume. A DESCRIPTION property is also created based on these properties.



	Property Name	Type	Value / Text Expression	Evaluated Value
1	MATERIAL	Text	"SW-Material@@@BASE PLATES@Conv	Material <not specified>
2	QUANTITY	Text	"QUANTITY@@@BASE PLATES@Convey	4
3	WEIGHT	Text	"SW-Mass@Conveyor Frame_L2_&.SLDP	368.795
4	DESCRIPTION	Text	PLATE, SW-Thickness x SW-Width x SW-	PLATE, 20 x 120 x 160
5	3D-Bounding Box Thickness	Text	"SW-3D-Bounding Box Thickness@@@	20
6	3D-Bounding Box Width	Text	"SW-3D-Bounding Box Width@@@BAS	120
7	3D-Bounding Box Length	Text	"SW-3D-Bounding Box Length@@@BA	160
8	3D-Bounding Box Volume	Text	"SW-3D-Bounding Box Volume@@@B	384000
9	<Type a new property>			

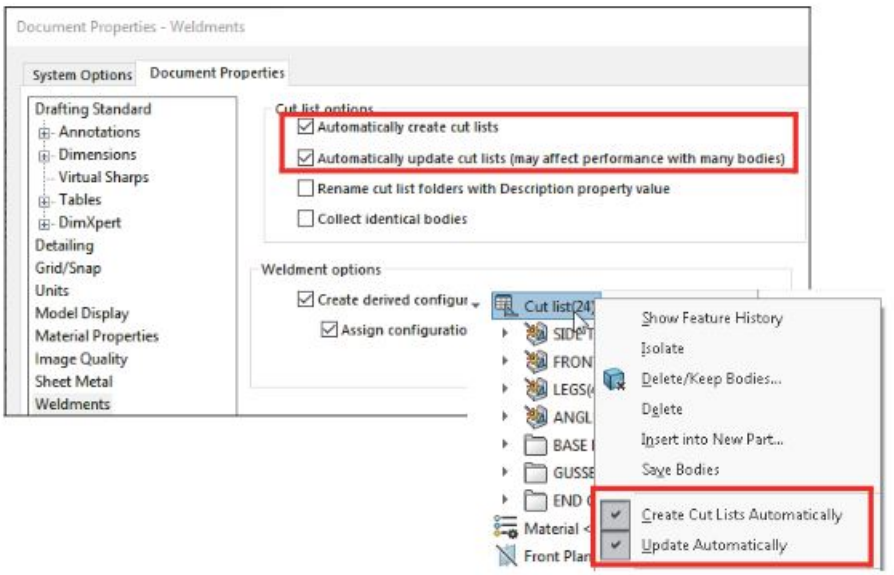
12 Add bounding boxes to the GUSSETS and END CAPS.



Create bounding boxes to add additional properties.

Options for Generating Cut List Items

Cut list items are generated according to the settings defined in Document Properties. By default, the settings to automatically create and update cut list items are turned *on*. This means SOLIDWORKS will group geometrically identical bodies into cut list item folders as they are created. These options can be adjusted if performance is being affected or if cut list items need to be created manually for non-identical bodies. These settings are accessible in the Options dialog as well as by right-clicking on the Cut list top level folder.



The option to **Create Cut List Automatically** must be turned on to allow the system to group bodies together to form cut list items.

The option to **Update Automatically** allows the system to group the bodies as they are created. By turning off the **Update Automatically** option, bodies will only be grouped into cut list items when prompted.

A **Refresh** symbol appears over the top level Cut list folder to indicate when an update is required. The cut list can be updated by right-clicking the top level Cut list folder and clicking **Update**.

Bodies that do not belong to a cut list item will not appear as line items in the cut list table.

Manually Managing Cut List Items

Cut list items can be created and managed manually when the automatic create and update options are turned off. To create a cut list item manually:

1. Select bodies listed in the **Cut list** folder, using **Shift** or **Ctrl** to group-select.
2. Right-click, and choose **Create Cut List Item** from the menu.

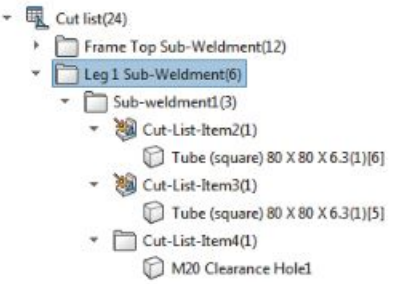
Bodies can also be dragged and dropped into existing cut list item folders.

Cut list items can be dissolved by deleting the cut list item folders.

Creating Sub-weldments

Sometimes it is necessary to break large weldments into smaller sections. Typically this is done to facilitate transportation. These smaller sections are called sub-weldments.

Sub-weldments are folders into which you put the associated solid bodies. The sub-weldment folder is then grouped into a cut list item folder and will appear as one line item cut list table.



The sub-weldment contains its own cut list items and can be saved as separate multibody part if desired. You can then make a drawing and cut list table for the sub-weldment.

To create a sub-weldment:

1. Select the bodies to include in the sub-weldment, using **Shift** or **Ctrl** to group-select.

Tip

To make selections in the graphics area, use the Selection Filter to **Filter Solid Bodies**. See *Using Selection Filters* on page 74 for more information.

2. Right-click and select **Create sub-weldment**.

   A **Sub-weldment Folder** containing the selected entities appears under the **Cut list** folder.

3. Update the **Cut list**, if needed.

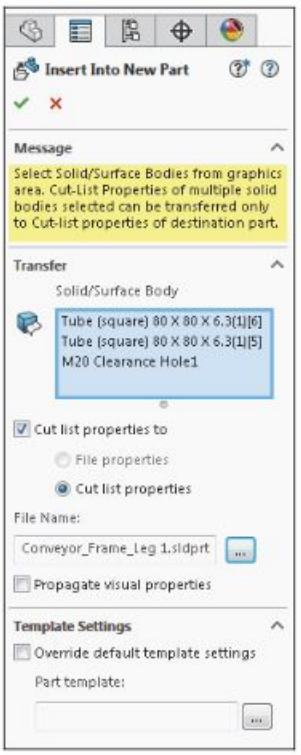
   If the Automatic update options are turned off, the `Cut list` may need to be updated to group the sub-weldment into a cut list item. It must be added to a cut list item to appear in the cut list table.

Tip

Sub-weldments can be dissolved by deleting the sub-weldment folder.

To save a sub-weldment to a new multibody part:

1. Right-click the **Sub-weldment Folder** and select **Insert into New Part**.
2. Use options in the PropertyManager to adjust settings and assign a file name and location.


Insert Into New Part
Message
Select Solid/Surface Bodies from graphics area. Cut-List Properties of multiple solid bodies selected can be transferred only to Cut-list properties of destination part.
Transfer
Solid/Surface Body
Tube (square) 80 X 80 X 6.3(1)[6]
Tube (square) 80 X 80 X 6.3(1)[5]
M20 Clearance Hole1
Cut list properties to
File properties
Cut list properties
File Name:
Conveyor_Frame_Leg 1.sldprt
Propagate visual properties
Template Settings
Override default template settings
Part template:

Any body can be saved to a new part file using either **Insert into New Part** or the **Save Bodies** feature. For more information on using these commands, see the *SOLIDWORKS Advanced Part Modeling* course.

Linked Cut List Properties

When you create sub-weldments, or when you save weldment bodies to new parts, the cut list properties from the parents propagate to the sub-weldments or new parts.

In the Cut-List Properties dialog box, the **Value / Text Expression** of the properties appear as **Linked to Parent Part- "parent".sldprt**. You cannot edit the cut list properties unless you break the reference to the parent part.

Using Selection Filters

When performing some operations in weldments, like creating sub-weldments, it can be useful to select solid bodies from the graphics area. By default, faces, edges, and vertices can be selected directly on a model. To control what is being selected when clicking in the graphics area, selection filters can be used. There are several default keyboard shortcuts associated with selection filters.

Keyboard Shortcut	Behavior
F5	Toggle Selection Filter Toolbar to be shown and hidden
F6	Toggle last selected filter on and off
X	Toggle selection filter for Faces on and off
E	Toggle selection filter for Edges on and off
V	Toggle selection filter for Vertices on and off

Selection filters can also be accessed from the **Context Toolbar** by clicking the **Selection Filter** flyout.

The cursor displays as when a selection filter is active.

A solid bodies selection filter can be activated to allow entire bodies to be selected directly from the model. Alternatively, solid bodies can always be selected from the `Solid Bodies` or `Cut List` folder in the FeatureManager design tree.

Tip

If using the solid bodies selection filter frequently, consider creating a custom keyboard shortcut or adding the command to a visible toolbar. These actions can be done through the `Customize` dialog (**Tools, Customize**).

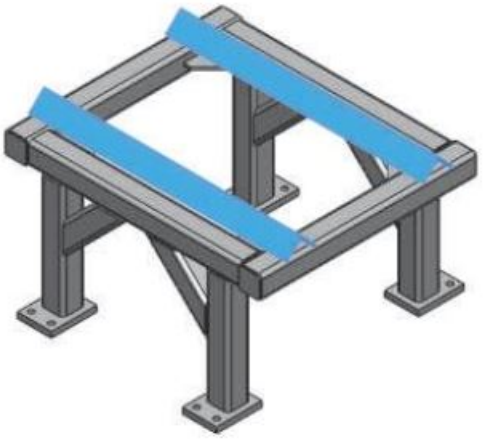
Custom Structural Member Profiles

Custom profiles can be created for use within the structural member feature. These profiles may be created by modifying existing profiles, creating a new profile from scratch, or by downloading available profiles from sources such as 3DContentCentral.

As stated in the previous lesson (page 14), Structural Member profiles are simply a 2D, closed contour sketch that is saved as a library feature part (*.sldlfp). In order for the library feature part to be used as a structural member profile, it must be saved to a Weldment Profiles folder location as defined in Options (page 16).

Modifying a Profile

The `Conveyor Frame` model requires some additional structural members that will need a modified profile.



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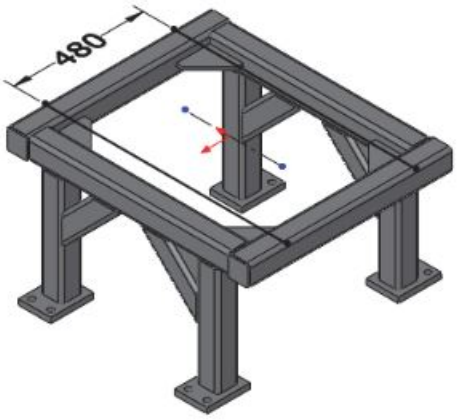
13 Sketch.
Select the reference plane `Top of Frame` and insert a sketch. Sketch two lines as shown. Use mirroring or sketch relations to make them symmetrical with respect to the origin.



14 Exit the sketch.

15 Insert a structural member.
Click **Structural Member** .

Standard: `ISO_Training`

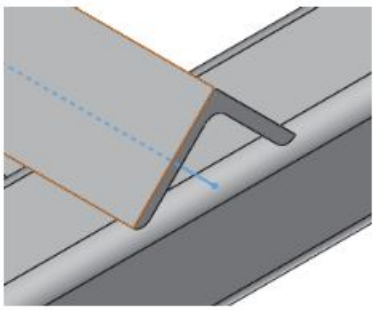
Type: `L Angle (equal)`

Size: `75x75x8`

Select both lines for `Group1`.

16 Locate Profile.

Click **Locate Profile**.



The L Angle should sit on the top of the frame with the sharp corner pointed up. It should also be centered on the lines created in the layout sketch.

The profile does not contain a point that would allow for this position, so a change is necessary.

Click **Cancel** × to close the PropertyManager.

17 Open the library part.

Click **Open**.

Set file type to **All Files (*.*)** and browse to the folder:

`SOLIDWORKS Training Files\Weldments\Weldment Profiles\ISO_Training\L Angle (equal)`.

Select the library feature part `75x75x8.sldlfp` and click **Open**.

Tip

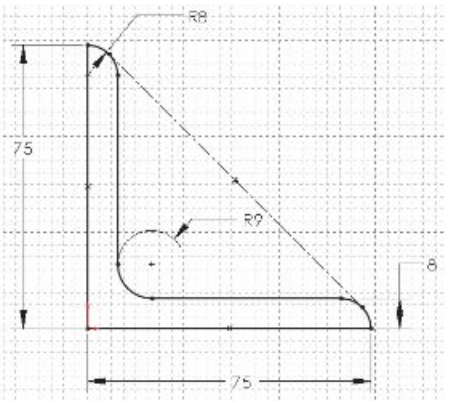
Windows Explorer can also be used to open Library Feature Parts (*.sldlfp). When opening files types that are not recognized as being associated with SOLIDWORKS, such as *.sldlfp, drag and drop them from Windows Explorer into the SOLIDWORKS application window.

18 Edit the sketch.

Sketch a centerline tangent to the two arcs.

Insert a point at the midpoint of the centerline.

Exit the sketch.



Transfered Information from Profiles

Structural member profiles can transfer common **Custom Properties** as well as **Material Definitions** to the cut list items and bodies that use the profile.

Custom Properties

The profile library parts should carry any custom properties that are commonly used, unique to the profile, and that should be imported into the cut list. For example, the profiles supplied with the software have a custom property named **Description**.

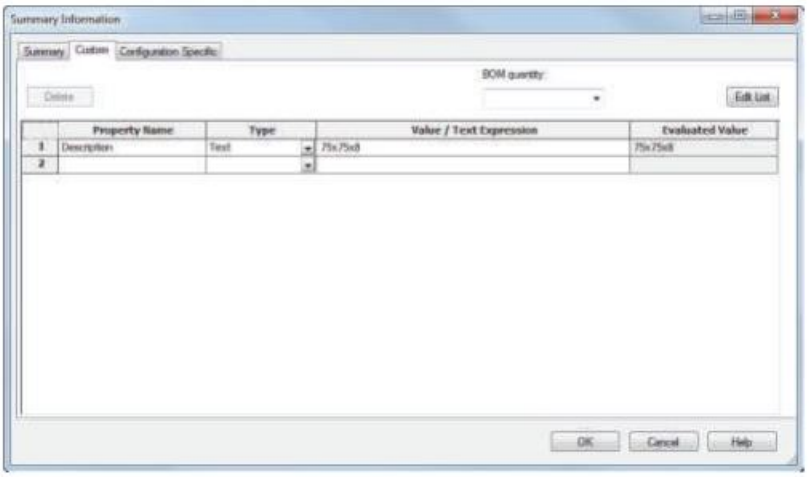
Material Definition

If a weldment profile is specific to a certain type of material, it can be added to the library feature part. To transfer the material definition to the bodies created in the structural member feature, use the option to Transfer Material from profile in the Structural Member PropertyManager.

19 Custom properties.

Click **File Properties** . Click the **Custom** tab.

Examine the information for the property name **DESCRIPTION**.



It is important to have a description associated with the profile because this custom property will be used when the cut list is generated. If any dimensions were changed in the sketch, you should update the description. No dimensions were changed here, so this description is still valid.

Click **OK** to close the dialog.

20 File, Save As.

Save the modified library part as `Modified_75x75x8.sldlfp`.

Close the library part.

21 Insert a structural member.

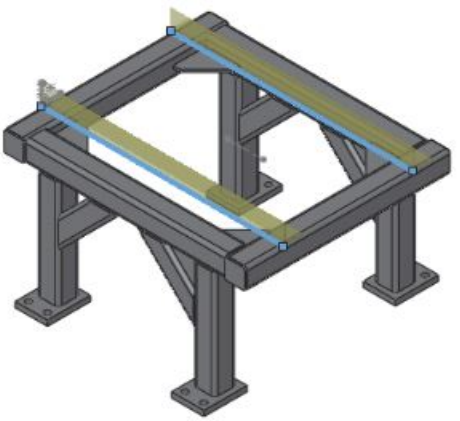
Click **Structural Member** .

Standard: `ISO_Training`

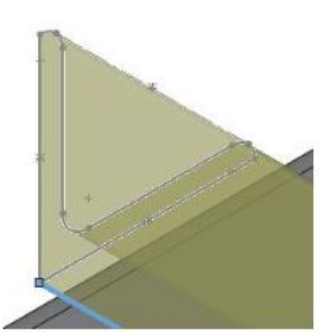
Type: `L Angle (equal)`

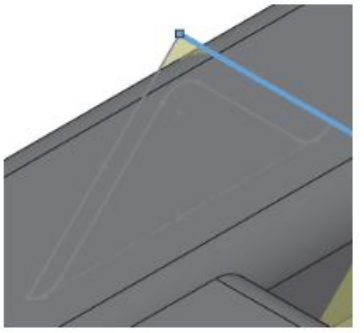
Size: `Modified_75x75x8`

Select the path segments.



22 Rotate the profile.
Under **Settings**, enter **225°** for the **Rotation Angle**.



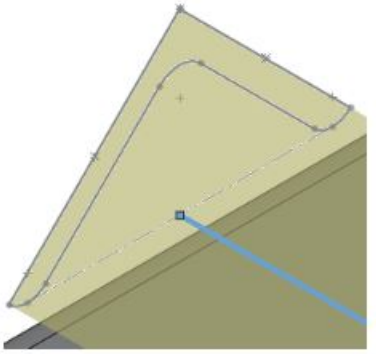


23 Locate Profile.
Click **Locate Profile.**

The system zooms in on the profile sketch.

Select the sketch point at the midpoint of the centerline.

Click **OK** ✔.



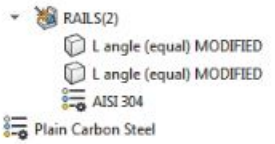
24 Rename the cut list item.
Rename the cut lit item RAILS.

Defining Material

Not all members in the weldment part have to be the same material. First, we will define a general material for the whole weldment. Then we will make changes to individual solid bodies.

25 Part material.
Right-click the Material<not specified> icon in the FeatureManager design tree and click **Plain Carbon Steel**. This material is applied to the entire weldment.

26 Material for individual bodies.
Expand the cut list item for the RAILS.



Use the **Ctrl** key to select both solid bodies. Right-click and choose **Material, Edit Material**.

Use the Material dialog to select **Steel, AISI 304.**

Click **Apply** then **Close**.

The rails are now specified as 304 stainless steel.

27 Review cut list properties.
Use the Cut-List Properties dialog to review the cut list properties.

The **MATERIAL** properties for each item is linked to the material of the solid bodies.

28 Save and close the part.

Creating Custom Profiles

Any 2D, closed contour sketch can be saved as a library feature part for use in structural member features. This provides a lot of flexibility for weldment model functions. Profiles can be created to represent bar and round stock, lumber sizes, gasket profiles, and more. Weldment functions can be used to produce virtually any type of model that can benefit from designing within the SOLIDWORKS multibody part environment and being described with a cut list.

In this next case study, the fence panel pictured will be created using a mixture of square tubing and bar stock. The bar stock profiles will need to be created in order to make those pieces as structural members.

1 **Open the existing part Fence Panel.**
Open the part Fence Panel from the Lesson02\Case Study folder.

2 **Add the fence posts structural members.**
Click **Structural Member** .

Use the following selections for the structural member profile:

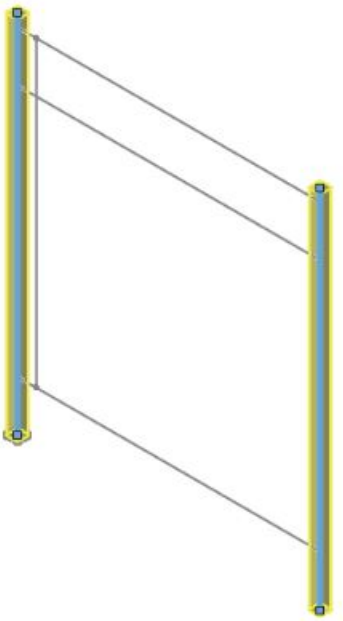
Standard: ISO_Training

Type: Tube (square)

Size: 80 x 80 x 6.3

Select the two vertical lines in the layout as shown.

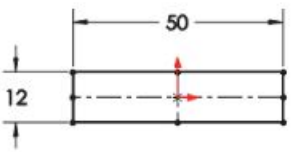
Click **OK** ✔.



3 **New part.**
The remaining members will be bar stock.

Start a new part document from the Part_MM template.

4 **Sketch profile.**
Sketch the profile as shown on the Front Plane.



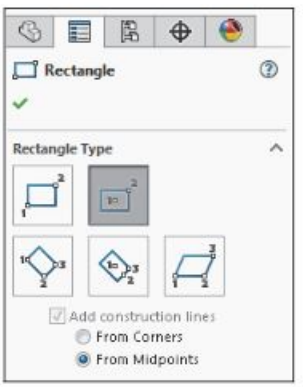
Note

The vertexes and endpoints in the profile are able to be used as pierce points when locating the profile. Sketch points can also be used to create additional pierce locations.

Tip

Use the option to sketch a center rectangle **From Midpoints** to automate the vertical and horizontal construction lines.

Exit Sketch .

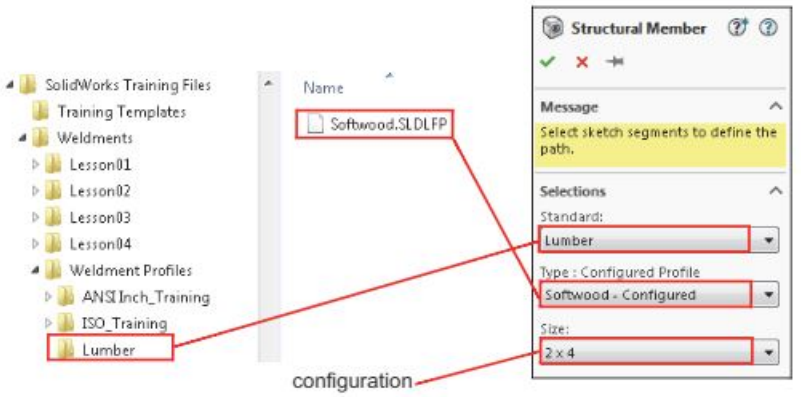


Standard or Configured Profiles

Before saving a custom profile, the decision needs to be made as to whether it will be a standard, stand-alone profile representing a single size, or whether it will be a configured profile containing multiple sizes. As stated in the previous lesson, the folder structure and naming conventions will differ based on this decision (see *Weldment Profiles Folder Structure* on page 17).

Since there are many sizes of bar stock that share this same simple contour, it makes sense to configure the profile to represent multiple sizes. Configurations can be easily added as needed and will ensure consistency with the profile contour and pierce locations.

When creating a configured profile, the name of the library feature part should reflect the *Type* of structural member and the configuration names should represent the *Size*. The folder the profile is saved to will appear as the *Standard* (see *Configured Profiles* on page 18).



---

5 Save as a library feature part.

Highlight `Sketch1` in the FeatureManager design tree.

Click **File > Save As** and change the **Save as type** field to **Lib Feat Part (*.sldlfp)**.

Browse to `SOLIDWORKS Training Files\Weldments\Weldment Profiles` and create a new folder called `Bar Stock`.

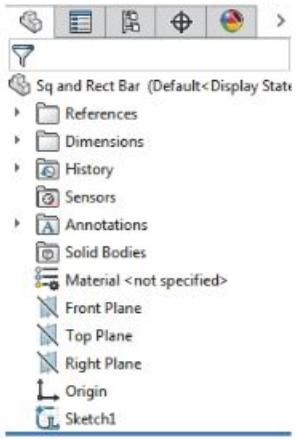
Name the file `Sq and Rect Bar` and save it into this folder.

Tip

It is good practice to store custom weldment profiles outside of the SOLIDWORKS installation directory. When performing a clean installation of SOLIDWORKS, the files in the installation directory will be deleted. If the weldment profiles are stored *outside* this directory, they will not be deleted and will still be available for use.

6 Results.

The active part is now `Sq and Rect Bar.SLDLFP`



The sketch is marked with a green "**L**" to indicate this is the feature that will be reused in the library part.

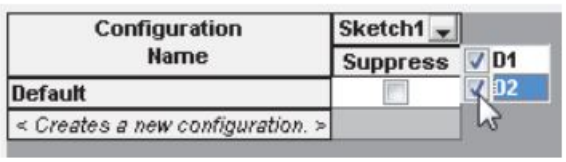
Tip

If the "**L**" is missing from the sketch icon, right-click the sketch and click **Add to Library.**

7 Configure the profile.

Right-click `Sketch1` and click **Configure Feature**.

In the dialog, use the drop-down menu to add the sketch dimensions as columns in the table.



8 Rename the dimensions and configuration name.

Right-click the dimension column headers and click **Rename**. Change the dimension names to `LENGTH` and `WIDTH`.

Right-click on the configuration row header and click **Rename Configuration**. Rename it `12 x 50mm`.

9 Create a new configuration.

Click in the last row header and type `20 x 20mm` as the name for the new configuration.

Type **20** for the `LENGTH` and `WIDTH` dimensions.

Configuration Name	Sketch1		
	Suppress	LENGTH	WIDTH
12 x 50mm	☐	50.000mm	12.000mm
20 x 20mm	☐	20.000mm	20.000mm
< Creates a new configuration. >			

10 Add Description properties.

Structural members inherit their description from the profile library part that is used. Since this is a configured profile, the Description will be a Configuration Specific Property.

Click **Hide/Show Custom Properties** in the dialog to show Configuration Specific Properties in the table.

Right-click the **New Property** column header and **Rename** to DESCRIPTION.

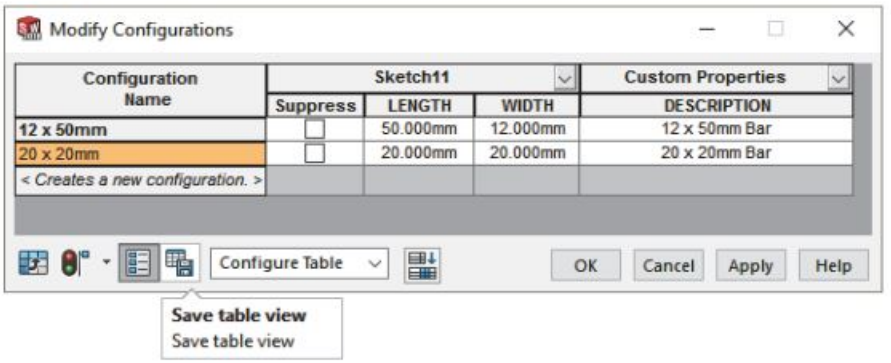
Add the descriptions `12 x 50mm Bar` and `20 x 20mm Bar`.

Click **Apply**.

Preview the configurations by double-clicking the row headers.

11 Save table view.

Give the table a name and click **Save table view**. This will store the table in the ConfigurationManager.



Click **OK**.

12 Save and close the profile library part.

13 Use the profile for a new structural member.

Click **Structural Member**.

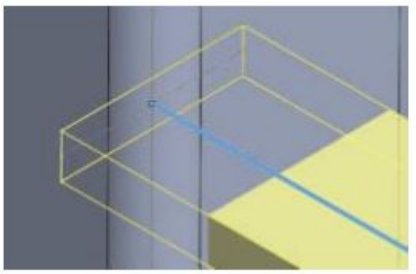
Use the following selections for the structural member profile:

Standard: `Bar Stock`

Type: `Sq and Rect Bar - Configured`

Size: `12 x 50mm`

Select the three horizontal lines in the layout.



Use settings to rotate the profile as shown.

Note

The default pierce location is the Origin of the profile sketch - the center of the rectangle.

Click **OK** ✔.

14 Add another structural member.

Click **Structural Member** .

Use the following selections for the structural member profile:

Standard: Bar Stock

Type: Sq and Rect Bar - Configured

Size: 20 x 20mm

Select the remaining vertical line in the layout sketch.

Click **OK** .

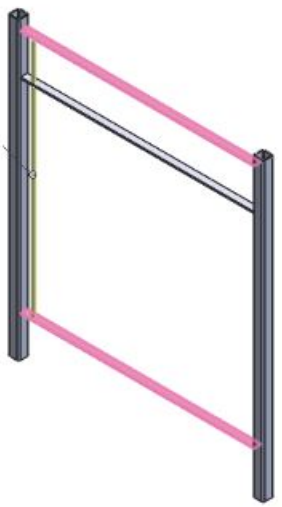
15 Trim the horizontal bars.

Use **Trim/Extend** to trim the horizontal bar bodies to the vertical posts.

16 Trim the vertical bar.

Use **Trim/Extend** to trim the vertical bar at the top and bottom.

Ignore the middle bar for now.



17 Pattern the vertical bar.

Create a **Linear Pattern** of the vertical bar body in the **X** direction.

Click **Up to reference** and select the inside face of the right fence post body.

Spacing: 114.3mm

18 Trim the center horizontal bar.

Use **Trim/Extend** to trim the middle horizontal bar around the vertical pieces.

Use a **Weld Gap** of **1mm** to add clearance.

19 Review the cut list.

Twenty (20) separate bodies in the model have been grouped into 4 cut list items.

Use the Cut-List Properties dialog to view the properties and preview the cut list table.

Close the dialog. **Save** the part.

Inserting Existing Parts

Existing parts can be inserted into a weldment model or any other part. Settings can be used to determine if a link is retained to the part being inserted. If the link is broken, all features transfer to the model being worked on.

The inserted part can be located using constraints or by translating and rotating. Settings allow the part to be located on insertion, or the **Move /Copy Body** command can be used at anytime. For more information on using **Insert Part**, refer to the *SOLIDWORKS Advanced Part Modeling* course.

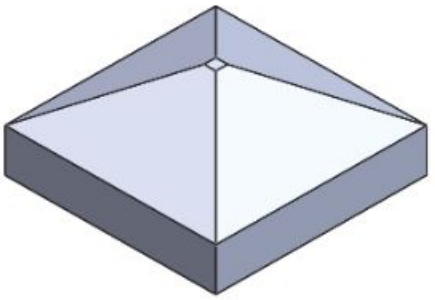
Where to Find It

- Menus: **Insert > Part**

---

20 Open the Fence Post Cap.

Open the existing part `Fence Post Cap` from the `Lesson02\Case Study` folder.



The part contains several configurations. Currently the `100x100` configuration is active.

Note

When inserting an existing part the current or last saved configuration will be used. We could switch it now, but will wait in order to demonstrate how to modify the configuration after the part has been inserted.

21 Activate the `Fence Panel` part.

Switch to the document window containing the `Fence Panel`.

22 Insert the Fence Post Cap.

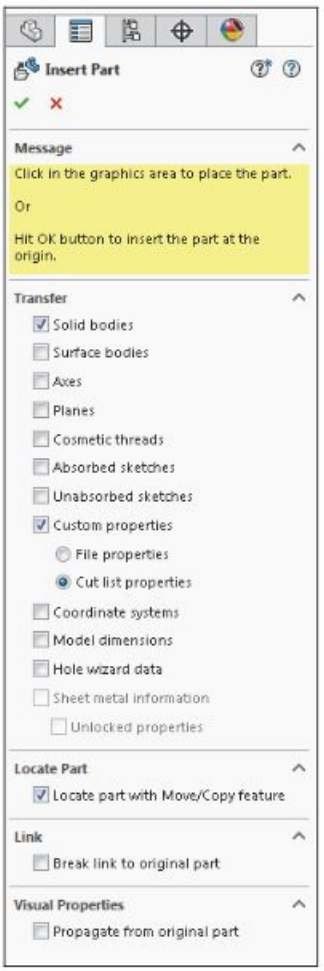
Click **Insert > Part**.

Select the Fence Post Cap and **Open**.

Use the PropertyManager to transfer the **Solid bodies** and **Custom Properties** as **Cut List Properties**.

Make sure the **Locate Part** option is checked. The **Break link** option should be cleared.

Click in the graphics area to place the part near the top of a post.



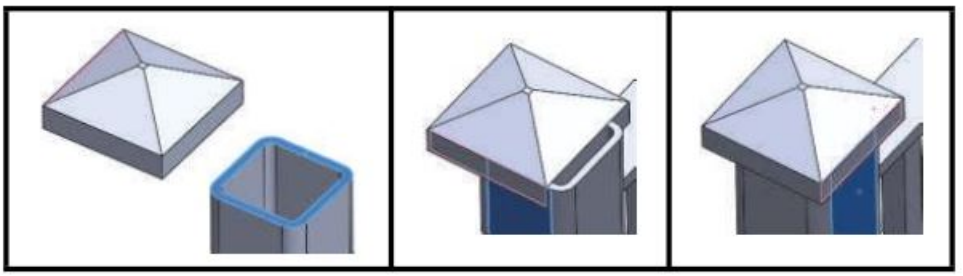
Note

Clicking **OK** in the PropertyManager drops the part at the Origin.

23 Locate part.

Use the **Locate Part Constraints** to position the cap.

Add **Coincident** mates to the *inside* edge and faces as shown.



Click **OK**.

Locate Part and Move/Copy Body

The **Locate Part** dialog can be accessed independently from the **Insert Part** feature. The **Move/Copy Body** command uses this same PropertyManager to move existing bodies to a new position or create new instances of a body at a specified location.

For more information on working with multibody parts including using **Insert Part** and **Move/Copy Body**, see the *SOLIDWORKS Advanced Part Modeling* course.

Where to Find It

- Menus: **Insert > Features > Move/Copy Body**

---

24 Results.

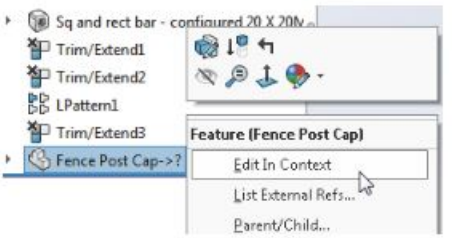
A new body and cut list item have been added to the model.

The custom properties from the inserted part have been transferred to the associated cut list item.

The part feature appears in the FeatureManager design tree.

The **->** symbol indicates the link to an external file. The configuration of the file that is being used is shown in parentheses (100x100).

Tip



If the symbol **->?** is shown, the reference is currently out of context. This means the referenced part is not open so the system cannot access the configuration information or geometry of the model. When out of context, the feature is displaying the referenced model as it was last known. Right-clicking an out-of-context feature provides an option to **Edit in Context**, which opens the part.

External references are covered in detail in the *SOLIDWORKS Assembly Modeling* course.

25 Switch configurations.
Right-click the Fence Post Cap feature.

Click **External Refs.**

Use the drop down menu in the dialog to change the active configuration name to 80x80.

Click **Close**.

26 Results.
The configuration and size of the body is updated.

The cut list item properties also update.

27 Mirror the Fence Post Cap body.

Mirror over the Right Plane to add another instance of the Fence Post Cap.

28 Cut List Properties.
Use the Cut List Properties dialog to view the cut list properties and preview the cut list table.

29 Save and close all files.

When to Use an Assembly

Even though it is possible to add existing parts to a weldment model, this technique should really only be done for standard part components or parts that can be described with the cut list. As a guideline, here are some things to consider when trying to decide if a component should be contained within the weldment model, or added as part of an assembly.

- Should the part be listed separately in a Bill of Materials (BOM) table?

  Weldments will appear as a single line item in upper level assemblies. If the component should have it's own line item in a BOM, it should be added as an assembly component.

- Does the part require it's own detail drawing?

  Complex models, such as castings, often cannot be fully described with a cut list. Although individual bodies can be detailed in a weldment, the drawing will be associated with the weldment part file (see *Drawing Views of Individual Bodies* on page 108). Standard or purchased parts can often be described with a part number and do not require a detail drawing.

- Should the pieces have some freedom of movement?

  The pieces of a multibody part are locked in position. Additional features can be use to move bodies within the part, but dynamic movement is not possible.

- Company standards must be considered.

  Some company standards may require that individual pieces of a weldment exist as a separate part and assembly documents for file management and part numbering purposes. SOLIDWORKS does include features such as **Save Bodies** which allow part and assembly documents to be generated from a multibody part, but these documents will be tied to the weldment "master model". See the *SOLIDWORKS Advanced Part Modeling* course for more information on saving solid bodies.

Exercise 5: Weld Table Cut List

Add material and create cut list item properties for the Weld Table.

This lab reinforces the following skills:

- *Cut List Item Names* on page 65
- *Defining Material* on page 79
- *Accessing Properties* on page 66
- *Adding Cut List Properties* on page 68
- *Bounding Boxes in Weldments* on page 69

Thanks to Adam Pohl, MJ Engineering

Units: **MMGS**

Procedure

1 Open an existing part.

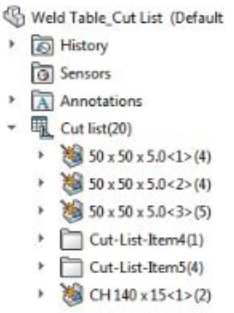
Use the `Weld Table` completed in *Exercise 2: Weld Table* on page 46 or open the part provided in the `Lesson02\Exercises` folder, `Weld Table_Cut List`.

The bodies of the part have automatically been grouped into cut list items.

2 Use the description property as the cut list item names.

Access **Options** ⚙, **Document Properties, Weldments** and select **Rename cut list folders with the Description property value**.

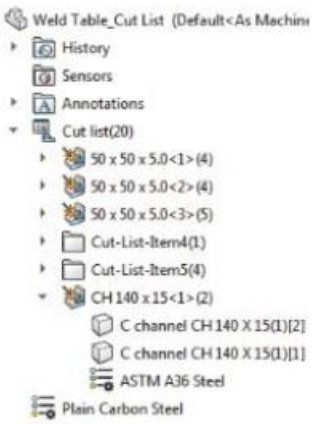
The cut list items created with structural member features inherit a description from the profile used. The other items will require a description property to be created for it to be used as the cut list folder name.



3 Add material.

Most members in the Weld Table are **Plain Carbon Steel**, so apply that material to the entire part.

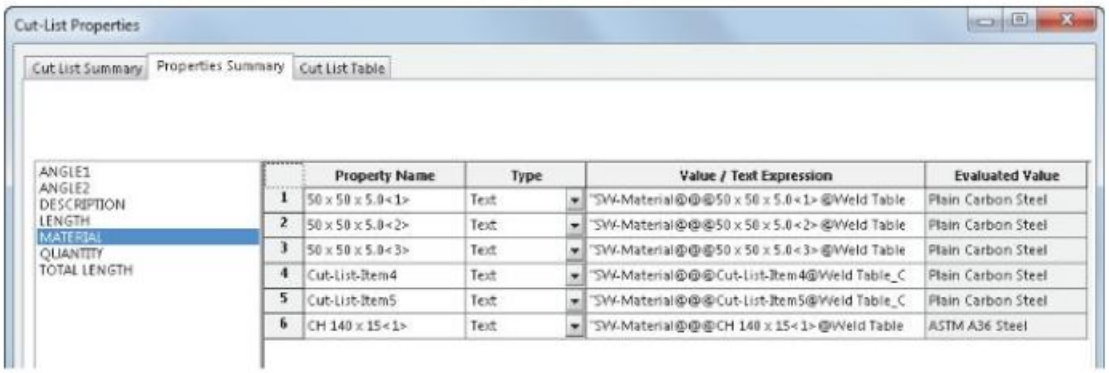
The channel members are **ASTM A36 Steel**. Add that material to the channel bodies.



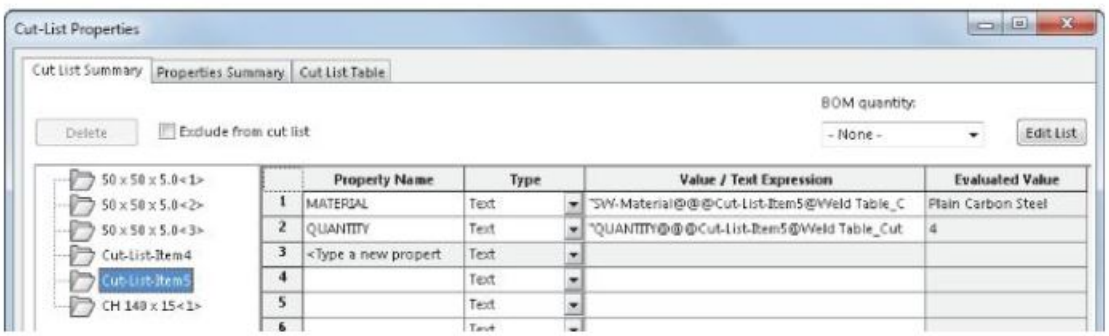
4 Preview Cut List Properties.

Right-click a cut list item folder and click **Properties** to preview the cut list item properties using the dialog.

The material property for each item it properly identified.


Cut-List Properties
Cut List Summary
Properties Summary
Cut List Table
ANGLE1
ANGLE2
DESCRIPTION
LENGTH
MATERIAL
QUANTITY
TOTAL LENGTH
Property Name
Type
Value / Text Expression
Evaluated Value
1 50 x 50 x 5.0<1> Text "SW-Material@@@50 x 50 x 5.0<1>@Weld Table Plain Carbon Steel
2 50 x 50 x 5.0<2> Text "SW-Material@@@50 x 50 x 5.0<2>@Weld Table Plain Carbon Steel
3 50 x 50 x 5.0<3> Text "SW-Material@@@50 x 50 x 5.0<3>@Weld Table Plain Carbon Steel
4 Cut-List-Item4 Text "SW-Material@@@Cut-List-Item4@Weld Table_C Plain Carbon Steel
5 Cut-List-Item5 Text "SW-Material@@@Cut-List-Item5@Weld Table_C Plain Carbon Steel
6 CH 140 x 15<1> Text "SW-Material@@@CH 140 x 15<1>@Weld Table ASTM A36 Steel

The cut list items for the base plates and table top currently only have the default MATERIAL and QUANITY properties.



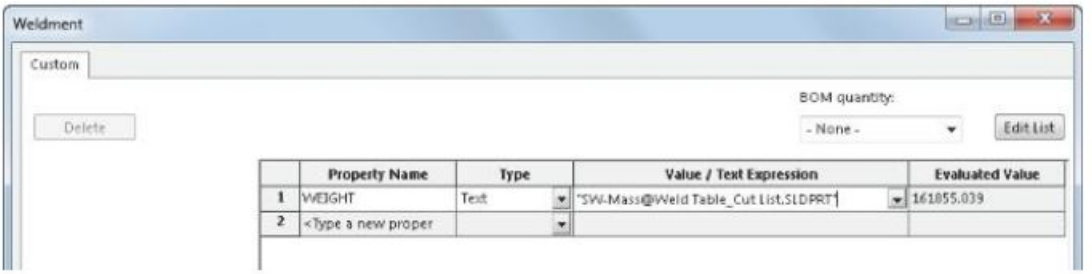
Click **OK** to close the dialog.

5 Add a WEIGHT property to all items.

Right-click the **Weldment** feature and click **Properties**.

Use the drop down menus in the dialog to add a **WEIGHT** property and link it to the **Mass** of each item.

Click **OK** to close the dialog.

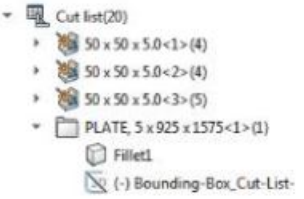


6 Create Bounding Box to add properties.

Right-click the cut list item folder for the table top.

Click **Create Bounding Box**.

A 3D Sketch and cut list properties are created. The **DESCRIPTION** property created is used as the cut list item name.

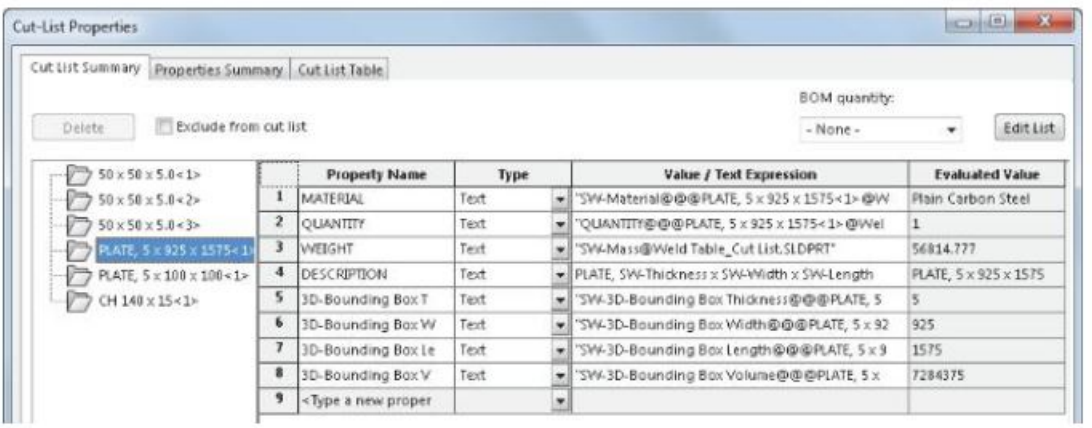


7 Create Bounding Box.

Add a Bounding Box to the remaining cut list item which contains the base plates.

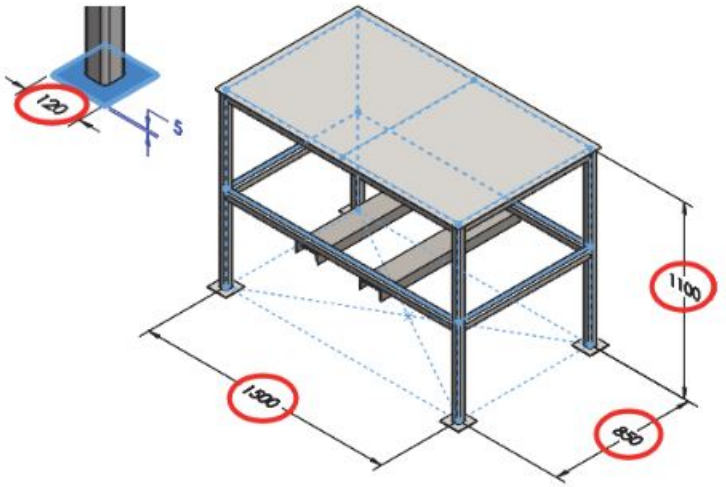
8 Cut List Properties.

Use the **Cut List Properties** dialog to evaluate the properties and preview the cut list table.



9 Make a change.

Change the dimensions of the Base Plate and Frame Layout as shown and **Rebuild** .



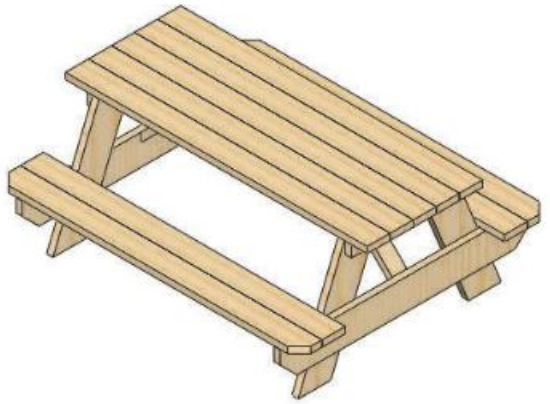
10 Updated properties.

Use the Cut List Properties dialog to review properties.

The size related properties for the cut list items will automatically update to the new dimensions.

11 Save and close all files.

Exercise 6: Picnic Table



Create a configured custom profile and use it to create the Picnic Table as a weldment model.

This lab reinforces the following skills:

- *Creating Custom Profiles* on page 79
- *Configure the profile.* on page 82
- *Save as a library feature part.* on page 81
- *Adding Cut List Properties* on page 68
- *Bounding Boxes in Weldments* on page 69

Dual Units: **IPS / MMGS**

Procedure

1 Open an existing part.
Open the `Picnic Table` part from the `Lesson02\Exercises` folder.

Although this model won't actually be welded, many woodworking projects can benefit from the flexibility of the weldment functions and can easily be described with the automatic cut list properties.

2 Modify configurations options.
The `Picnic Table` will not require the automated derived configurations.

In **Options** ⚙, **Document Properties**, select **Weldments.**

Clear the checkbox for **Create derived configurations**.

3 Required profiles.
The lumber sizes that are required for the `Picnic Table` are as follows (in inches):

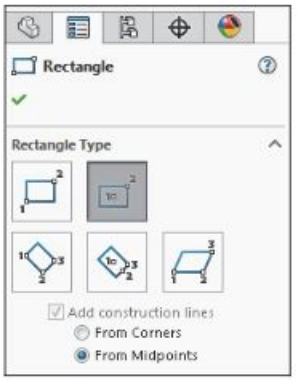
- 2 x 4
- 2 x 6
- 2 x 8

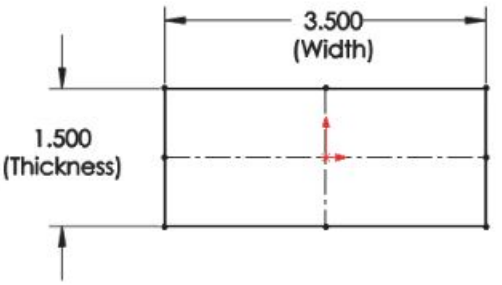
These similar profile sizes can easily be created within a single configured library part.

4 New part.
Begin a new part using the `Part_IN` template.

5 Create the 2 x 4 profile.

Use the Front Plane and a **Center Rectangle** to sketch the profile as shown.





Rename the dimensions to make them easy to identify when configuring the profile.

Exit Sketch.

6 Configure the profile.

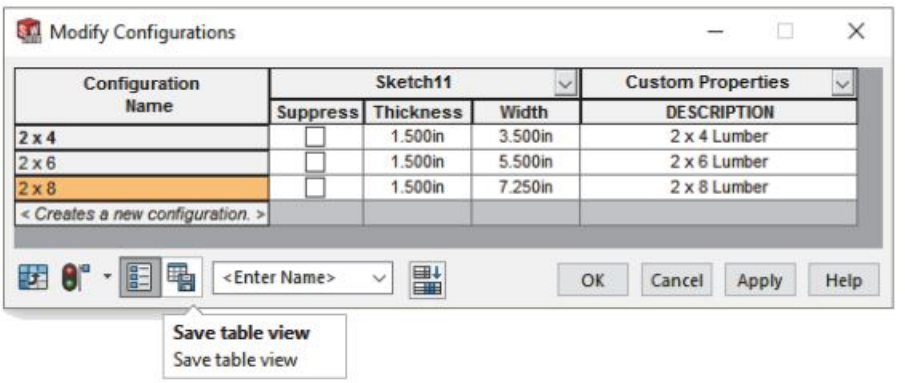
Right-click Sketch1 in the FeatureManager design tree and click **Configure Feature**.

Rename the Default configuration to be 2 x 4.

Use the dialog to add additional configurations and enter the values as shown.

Select the **Hide/Show Custom Properties** button in the dialog. Add a **DESCRIPTION** property as shown.

Name the table and **Save the Table View** so it is available to access at any time.



7 Preview configurations.
Click **Apply**. Double-click the configuration names to preview each in the graphics area.

Click **OK**.

Tip

The table is stored in the ConfigurationManager and can be accessed at any time by double-clicking it there.

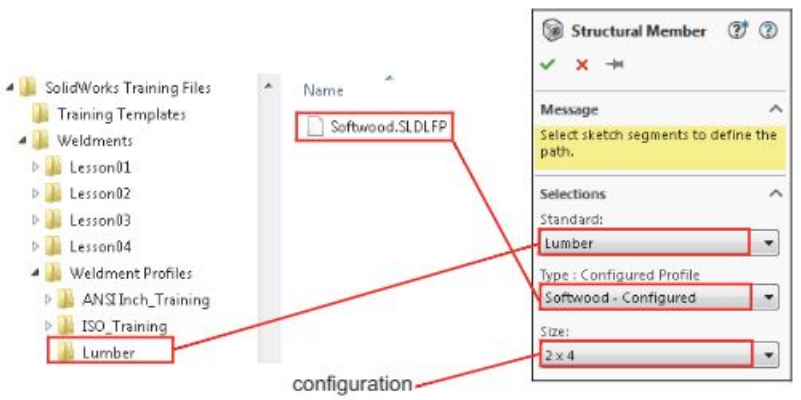
8 Save as library feature part.
To use the sketch as a profile it must be saved as a library feature.

Highlight Sketch1 in the FeatureManager design tree.

Click **File, Save As** and change the **Save as type** field to **Lib Feat Part (*.sldlfp)**.

Browse to SOLIDWORKS Training Files\Weldments\ Weldment Profiles and create a new folder called Lumber. For configured profiles, this folder name will be used as the *Standard*.

Name the file Softwood and save it to this folder. For configured profiles, the file name represents the *Type* of profile.



Verify the sketch shows the green "L" to indicate it is included as a library feature.

Close the Softwood library part.

9 Add 2 x 8 structural members.
In the Picnic Table part, click **Structural Member** .

Use the following selections for the structural member profile:

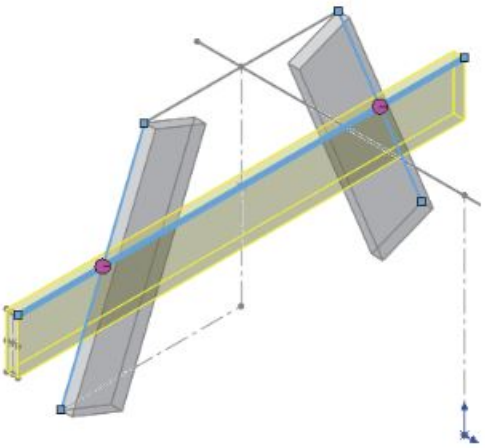
Standard: Lumber

Type: Softwood - Configured

Size: 2 x 8

Create the 3 groups as shown. Locating the profile as appropriate for each.

Click **OK** ✔.



10 Add 2 x 4 structural members.

Click **Structural Member** .

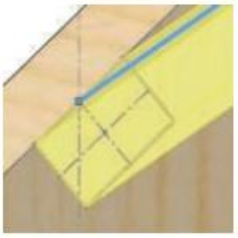
Use the following selections for the structural member profile:

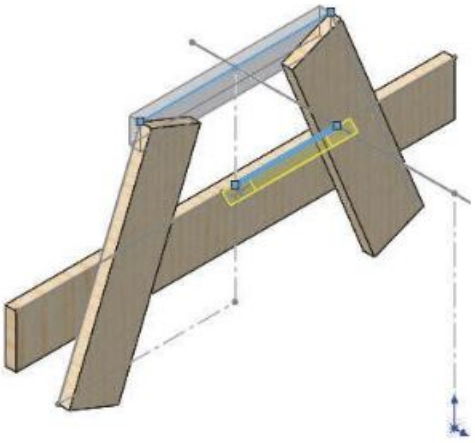
Standard: Lumber

Type: Softwood - Configured

Size: 2 x 4

Create 2 groups as shown. Use **Locate Profile** and settings to position the profiles.





11 Add 2 x 6 structural members.

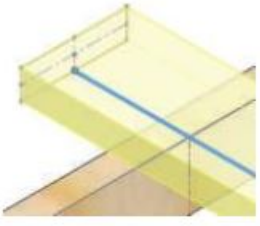
Use the following selections for the structural member profile:

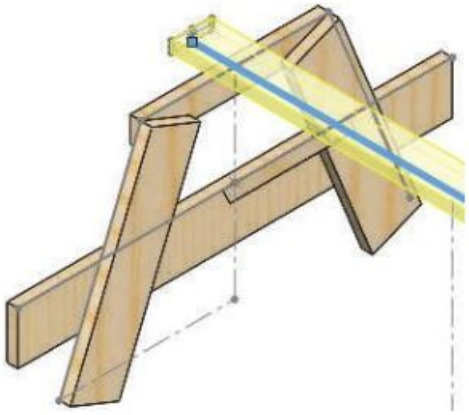
Standard: Lumber

Type: Softwood - Configured

Size: 2 x 6

Position the profile as shown.



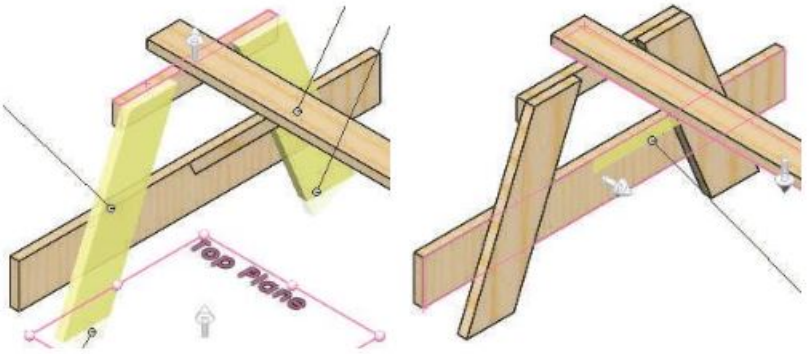


12 Trim/Extend members.

Hide the layout sketches.

Trim/Extend the side members of the A-Frame. For the **Trimming Boundary** click the **Face/ Plane** option and use the Top Plane and face as shown.

Trim/Extend the angled brace member to the faces as shown.



13 Mirror bodies.

Use the Right Plane to **Mirror** the bodies of the A-Frame and Brace.

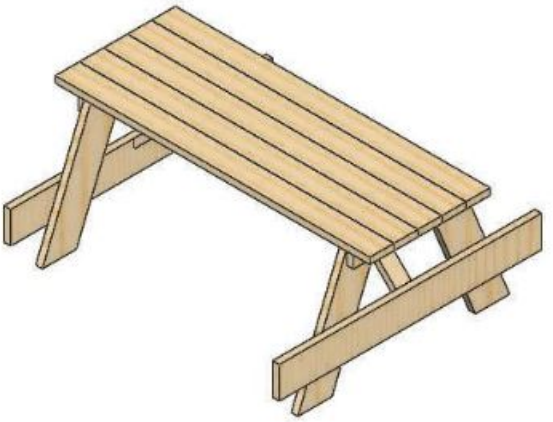
14 Pattern bodies.

Create a **Linear Pattern** of the top member in 2 directions using the following settings:

Spacing: 5.5

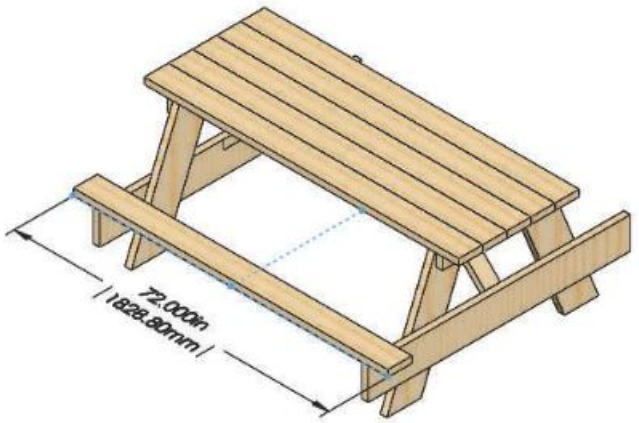
Instances: 3

Direction 2: Pattern seed only.



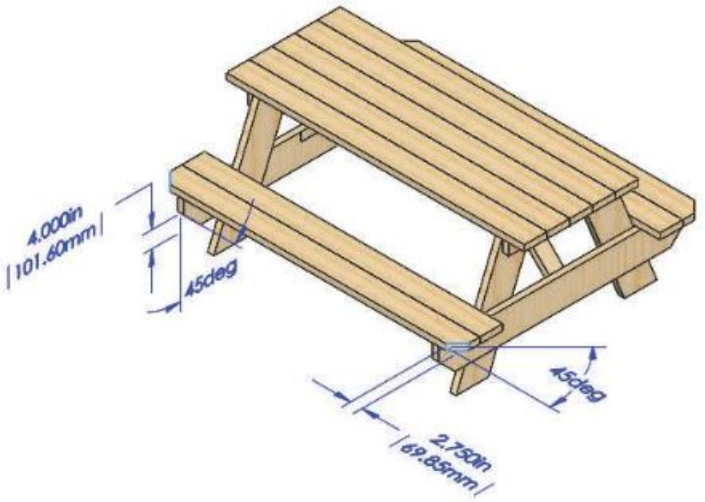
15 Create bench structural member.
Sketch the layout for the bench structural member using an existing face.

Use a **2 x 6** profile and position as shown.



16 Pattern bodies.
Use pattern and mirror features to complete the benches.

17 Optional: Chamfers.
Use the rollback bar to add chamfers to the bodies where appropriate - before patterning.



18 Cut List Properties.
Use the **Cut List Properties** dialog to view the properties for each item and preview the cut list table.

19 Save and close all files.

Exercise 7: Insert Part

Add welded rings to the Suspension Frame by inserting an existing part model.

This lab reinforces the following skills:

- *Creating Custom Profiles* on page 79
- *Configure the profile.* on page 82
- *Save as a library feature part.* on page 81
- *Adding Cut List Properties* on page 68
- *Bounding Boxes in Weldments* on page 69

Units: **MMGS**

Procedure

1 Open an existing part.

Use the Suspension Frame completed in *Exercise 3: Suspension Frame* on page 51, or open the part provided in the Lesson02\Exercises folder, Suspension Frame_Insert Part.

2 Insert Part.

Click **Insert > Part**.

Browse to the Lesson02\Exercises folder and open the Welded Ring.

Use the following settings in the PropertyManager:

Transfer: Solid bodies, Planes, and **Custom Properties** as **Cut list properties**

Locate Part: Checked

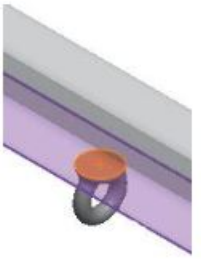
Link: Cleared

Click in the graphics area to place the part. Click **Yes** to the message regarding units if necessary.

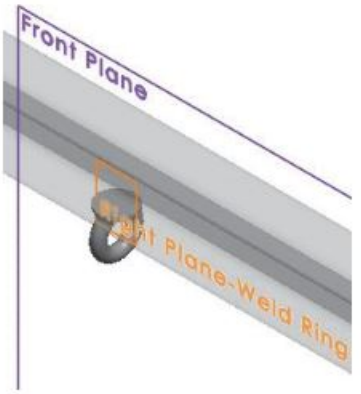
3 Locate Part.

Add the following constraints to position the Welded Ring (the Beam member is shown as transparent for clarity):

The **bottom face** of the Welded Ring is **Coincident** with the bottom face of the Beam.

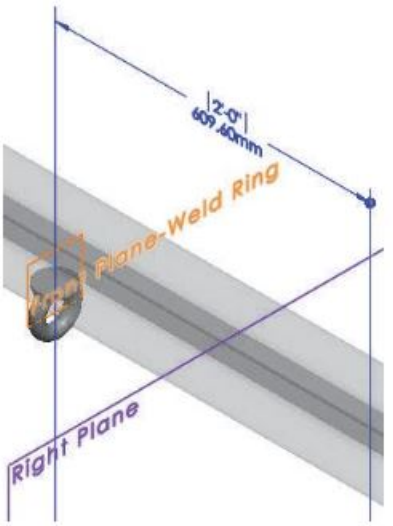


The Welded Ring Right Plane is **Coincident** to the Suspension Frame Front Plane.



The Welded Ring Front Plane is a **Distance** of **609.6mm** from the Suspension Frame Right Plane.

Click **OK** to complete locating the part.



Tip

If you need to modify **Locate Part Constraints** after they've been created, expand the part feature and edit the Body-Move/Copy feature listed.

4 **Mirror the body.**

Mirror across the Right Plane to add another instance of the Welded Ring.

5 **Cut List Properties.**

Use the Cut List Properties dialog to review the properties and preview the cut list table.

6 **Optional: Add material and properties.**

Define the material for the Suspension Frame to be **ASTM A36 Steel**.

For the base plates cut list item, add additional properties linked to the size of the feature.

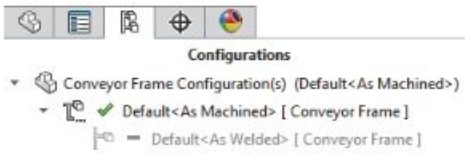
7 **Save and close all files.**

Lesson 3
Configuring and Detailing Weldments

Upon successful completion of this lesson, you will be able to:

- Represent post-assembly machining operations.
- Use Feature Scope settings.
- Represent weld beads.
- Create drawing views of weldment models and individual bodies.
- Create a cut list table in a drawing.
- Create a weld table in a drawing.

Weldment Configurations



The default settings for weldments are such that configurations are automatically created to represent the model in an "As Welded" state and an "As Machined" state. Using these configurations provides an easy way to communicate information about a weldment model in different stages of the manufacturing process.

Here are some basics on using weldment configurations:

- Derived configurations, such as the "welded" configuration, share a parent/child relationship with the top level configuration. This means if a change is made to the parent configuration it automatically transfers to the derived configuration. If a change is made when the *derived* configuration is active however, it will *not* automatically be represented in the parent.
- Features that represent post-assembly machining features should be suppressed in the As Welded configuration.
- The cut list properties will represent the As Welded configuration by default.
- When creating drawing views of a weldment model, use the drawing view properties to control which configuration is shown.

Note

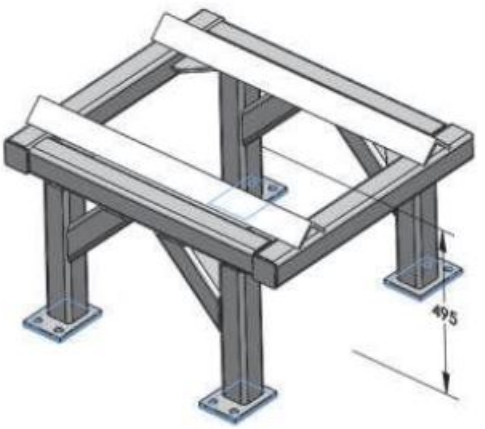
Options can be adjusted to prevent the creation of configurations if appropriate. See *Weldment Configuration Options* on page 10 for more information.

Adding Configurations

When the weldment options are set to create configurations, any new top level configuration that is added will automatically have a derived "welded" configuration associated with it.

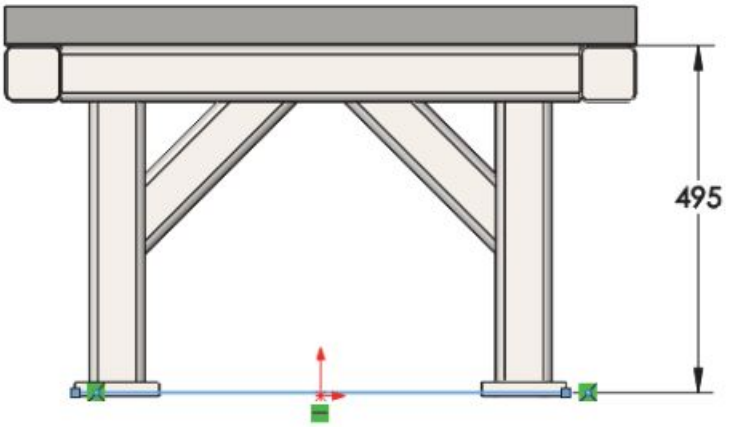
Post-Assembly Machining Features

Post-assembly machining features are generally holes and cuts added to a product after it has been assembled or welded. In the case of the Conveyor Frame, a cut will be made to level the Base Plates and ensure they are flush. The weldment configurations will be used to represent the model before and after machining.



1 **Open the Conveyor Frame.**
Continue working with the model previously created, or open the model provided in the Lesson03\Case Study folder, Conveyor Frame_L3.

2 **Sketch on the Front Plane.**
Create a the sketch as shown. Dimension to the Top of Frame plane.



3 **Extrude Cut.**
Click **Extruded Cut** . For **Direction 1** choose **Through All - Both**.

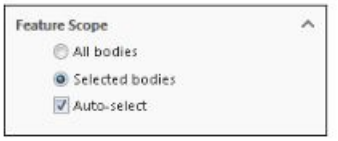
Select **Flip side to cut**, if necessary.

For **Feature Scope**, verify **Auto-select** is checked.

Click **OK** ✔.

Feature Scope

When working with a multibody part, certain features will include **Feature Scope** options. **Feature Scope** controls which bodies in the part will be affected by the feature.



- **Auto-select (default)**
  When Auto-select is checked, all visible bodies crossed by the feature will be selected as the affected bodies.
- **Selected bodies**
  When Auto-select is cleared, bodies can be manually selected from the part. Only selected bodies will be affected by the feature, all other bodies are ignored.
- **All bodies**
  All solid bodies, including hidden bodies will be affected by the feature.

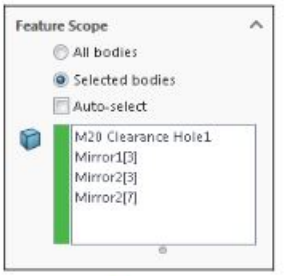
Once a feature has been created the feature scope can be modified by selecting or removing bodies from an available selection box in the feature PropertyManager.

---

4 Edit Cut-Extrude1.

Select the cut feature and click **Edit Feature**.

The bodies affected by the cut are now shown as the **Selected Bodies** in **Feature Scope**.



Click **OK** ✔.

5 Configure feature.

Right-click Cut-Extrude1 and click **Configure Feature**.

Suppress the cut in the Default<As Welded> configuration. As a post-assembly machining feature, this will not be present when the Conveyor Frame is being welded.

Configuration Name	Cut-Extrude1 Suppress
Default<As Machined>	☐
Default<As Welded>	☑
< Creates a new configuration. >	

Click **OK**.

Configurations can be previewed from the ConfigurationManager.

6 Save the part.

Weldment Drawings

Making a drawing of a weldment is the same as making a drawing of any other part but may require some unique detailing techniques such as:

- Adding views of individual solid bodies.
- Representing welds.
- Adding a cut list table or weld table.

At this point, a detail drawing of the `Conveyor Frame` will be created using all of these techniques.

Note

This section is not intended to be a comprehensive treatment of drawing production. The intent is only to illustrate topics that are unique to weldments. For a thorough discussion of creating engineering drawing, consider attending the *SOLIDWORKS Drawings* course.

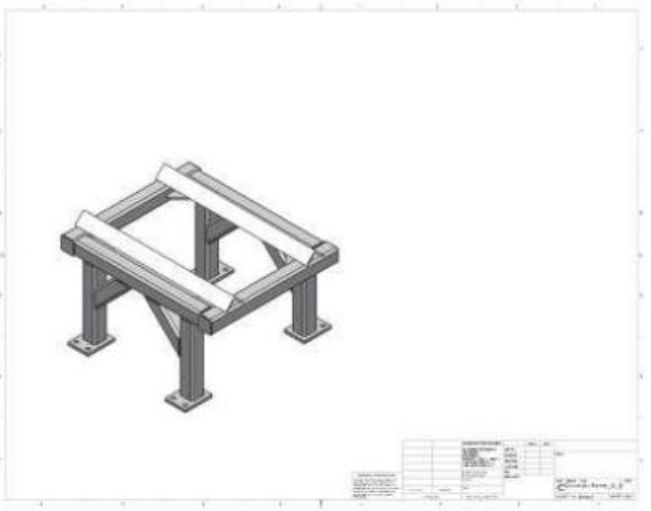
---

1 **Activate the `Default<As Welded>` configuration.**
Use the ConfigurationManager to activate the `Default<As Welded>` configuration. The first drawing sheet will detail this version of the model. The active configuration will be the default referenced configuration of drawing views created.

2 **Create a drawing.**
Click **Make Drawing from Part/Assembly** on the Standard toolbar. Use the `C_Size_ANSI_MM` template.

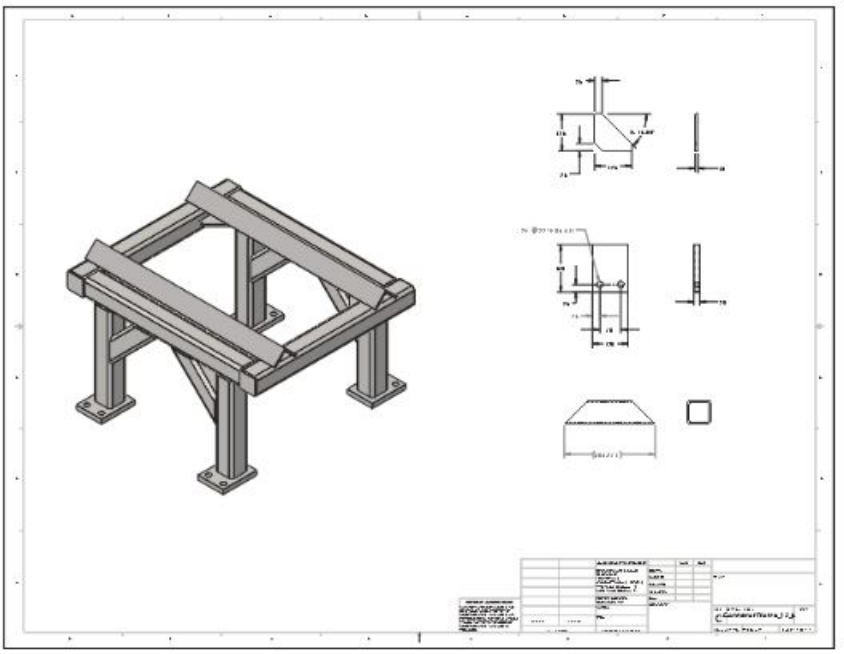
3 **Create a drawing view.**
Create an **Isometric** view. Make sure the referenced configuration is `Default<As Welded>`. Click **Shaded with Edges**.

4 **Edit the sheet scale.**
Edit **Sheet Properties**. Change the **Sheet Scale** to **1:5.**



Drawing Views of Individual Bodies

Most of the information required for producing the pieces of the Conveyor Frame will be communicated through the cut list table. However some pieces need some additional information to fully describe them.



To create a drawing view of an individual body, there are several options:

- **Select Bodies** to be included in an existing drawing view.
- Create a **Display State** in the model and use it in a standard drawing view.
- Create a **Relative View** in the drawing.

Select Drawing View Bodies

After creating a drawing view, the option to **Select Bodies** can be used to pick bodies to be included in the view. All other bodies will be hidden.

This technique will be used to add a view of the gusset.

Where to Find It

- Drawing View Properties: Select a drawing view, click **Select Bodies**

---

5 Create a Top view.
Add a **Top** view of the Conveyor Frame.

Tip

If using the **View Palette**, consider clearing the option to **Auto-start projected view** when creating a singular view.

6 Select Bodies.
Select the **Top** view.

In the **Drawing View** PropertyManager click **Select Bodies**. The model becomes active to make selections.

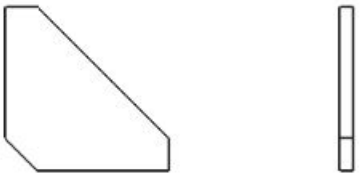
Click the `Gusset1` body; the original gusset created, not a mirrored instance.

Note

The original body created has dimensions associated with it, while patterned instances do not.

Click **OK** ✔.

7 Project View.
Create a **Projected View** to the right.



8 Optional: Insert Model Items.
From the Annotation toolbar, click **Model Items**.

Use the following settings:

Source: Selected Feature

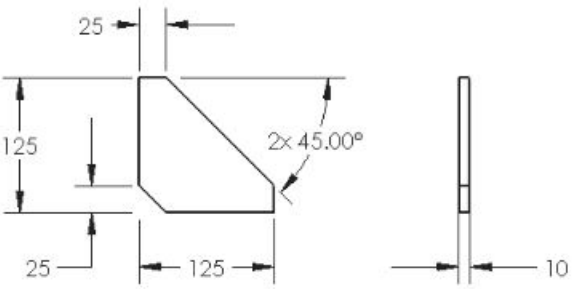
Destination View(s): Gusset drawing views

Dimensions: Marked for Drawings

Click a face of `Gusset1` to add dimensions of the feature to the drawing view.

Click **OK** ✔.

9 Optional: Modify dimensions.
Modify dimensions as appropriate.



Using Display States

Display States control the visual properties of a model and can be selected for a drawing view. Creating a display state with only a selected body visible allows a drawing view to be created of that individual body. Display States are stored in the ConfigurationManager and can be linked with configurations of the model or available for all configurations.

When using this technique, be sure to review existing display states if modifications are made to a model. New bodies added to the part will be shown in display states, so they may require updating.

For the `Conveyor Frame`, a display state will be used to create a view of the base plate.

Where to Find It

- ConfigurationManager: Right-click, click **Add Display State**, modify visual properties of the model
- **Isolate** command: Select bodies or components, right-click and select **Isolate**, click **Save as Display State**

Isolate

The **Isolate** command can be used in both multibody parts and assemblies. It allows for selected bodies or components to be isolated from other items in a model. All non-selected items will be hidden from view or can be turned transparent. When items are isolated, a dialog appears to control the visibility of removed bodies or components, save a display state, and Exit Isolate.

Clicking **Exit Isolate** will return to the previous visibility state of all items.

When working in an isolated state, additional bodies or components can be shown or hidden using the FeatureManager design tree or graphics area.

Where to Find It

- Shortcut Menu: Select geometry from bodies or components, right-click, **Isolate**

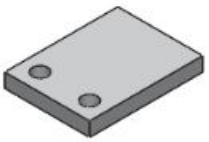
---

10 Open the `Conveyor Frame` model.

Activate the `Conveyor Frame` document window or open it from a drawing view.

11 Isolate the base plate.

Click a face of the original base plate body; the instance created with a sketch.



Right-click and select **Isolate**.

Tip

If a command is not directly visible in the shortcut menu, use the double arrows to expand the menu. Shortcut menus can be customized to choose which commands appear by default by clicking **Customize Menu** at the bottom of the commands list.

12 Save Display State.

Click **Save as Display State**.

Name the display state `Base Plate Detail`.

13 Exit Isolate.

14 Results.

The new display state can be accessed in the ConfigurationManager. Double-clicking a display state will activate it.

Note

The `Conveyor Frame` display states are *not* linked to configurations, so each display state is available in all configurations. This setting in found in the **Display State Properties** (right-click the display state, click **Properties**).

15 Save the part.

16 Create a drawing view of the base plate.

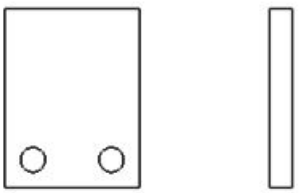
Activate the drawing document window.

Create a **Top** view and adjust the drawing view properties to use the `Base Plate Detail` display state.

Verify the `Default<As Welded>` configuration is being referenced.

17 Project View.

Create a **Projected View** to the right.



Note

With this technique, the drawing view border size still reflects the entire model. A projected view will need to be dragged past the border to be placed. To reduce the drawing border size, a **Crop View** can be used.

Tip When drawing view borders extend over the edges of the sheet, use **Zoom to Sheet** rather that **Zoom to Fit** to center the drawing sheet on screen.

18 Optional: Insert Model Items.

From the Annotations toolbar, click **Model Items**.

Use the following settings:

Source: Selected Feature

Destination View(s): Base Plate drawing views

Dimensions: Marked for Drawings, **Hole Wizard Location**, **Hole callout**.

Click a face of `Boss-Extrude1` and an edge of the hole to add dimensions for those features to the drawing views.

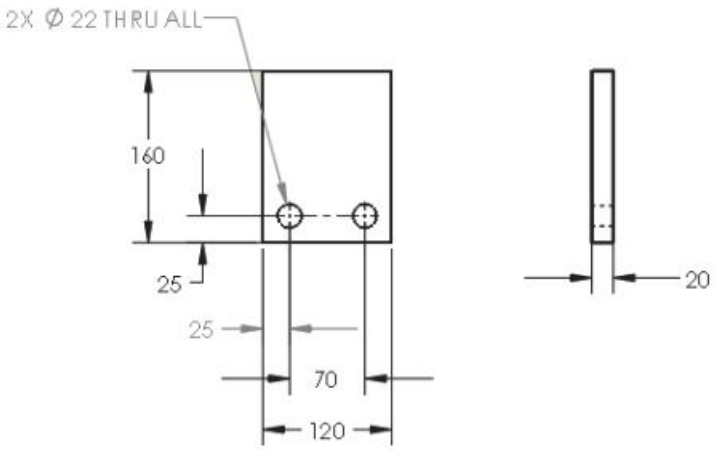
Tip Use the tool tips that appear as the cursor hovers over geometry to identify which feature a selection is associated with.

Click **OK**.

19 Optional: Modify dimensions and add annotations.

Modify dimensions by hiding, moving, and adding as appropriate.

Add **Center Marks** and **Centerlines**.



Using Relative View

Relative Views create a drawing view of a model and orient it relative to selected faces or planes. Options within the **Relative View** command allow for the view to be based off the entire model or only selected bodies. **Relative Views** are useful when there is not an existing view orientation available to achieve the desired view of a model or solid body.

Where to Find It

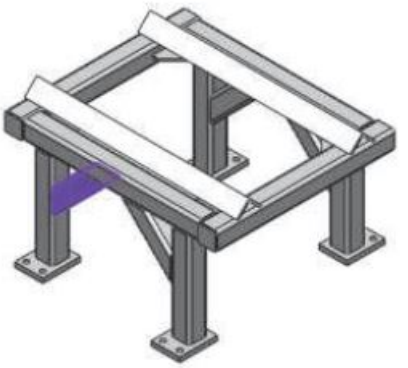
- Shortcut Menu: Right-click a drawing view, click **Drawing Views, Relative View**
- Menu: **Insert, Drawing View, Relative To Model**

---

20 Relative view.
Click **Relative View**.

The model document becomes available for making selections.

For **Scope**, click **Selected Bodies** and select the angled brace as shown.



21 Orientation.
Make sure **First orientation** is set to **Front** and select the front face of the angled brace.

Set **Second orientation** to **Bottom**. Rotate the view and select the face shown.

Click **OK** ✔.

The drawing document window becomes available for locating the view.

22 Place view.
Click on the drawing sheet to position the view.

23 Drawing view properties.
Modify the drawing view properties to be **Hidden Lines Visible** and **Use sheet scale.**

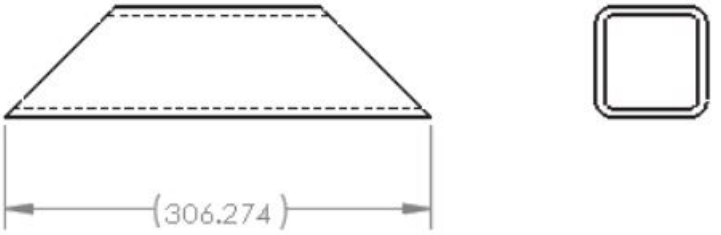
Right-click the relative view and click **Tangent Edge, Tangent Edges Removed.**

24 Create a Project View.

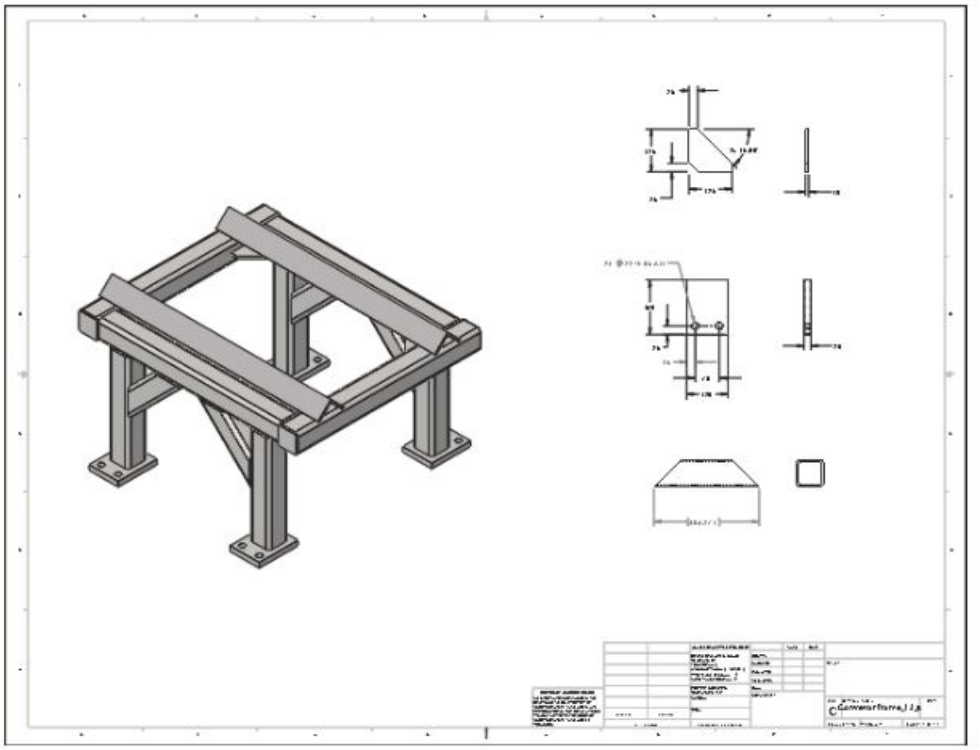
Create a **Projected View** to the right of the relative view.

25 Optional: Add a dimension.

Add a dimension to show the length of the angled brace.



26 Save the drawing.



Cut List Tables

Insert a cut list table to create a list of the components in the weldment. The properties associated with the cut list items are imported into the table.

Where to Find It

- CommandManager: **Annotations > Tables > Weldment Cut List**
- Menu: **Insert, Tables, Weldment Cut List**

27 Cut list table.

Click **Weldment Cut List** and select the isometric view of the weldment.

The **Table Template** will be `cut list`.

Note

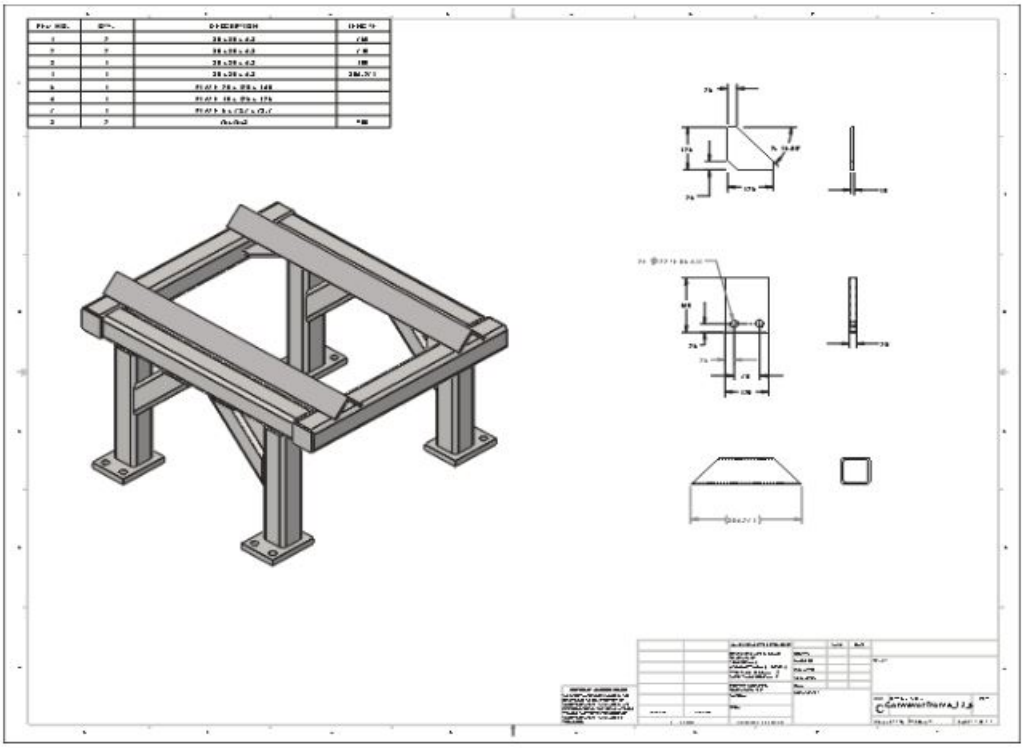
If this is not the template listed, click the **Browse Template** button and select `cut list.sldwldtbt` from the installation directory.

Under **Table Position**, select the **Attach to anchor point** check box.

The table is linked to the `Default<As Welded>` configuration.

Click **OK** .

28 Results.



Cut List Table Settings

The initial settings of a cut list table can be controlled in 2 places:

- Drawing Document Properties
- Cut List Table template

In **Options > Document Properties > Tables**, the setting **Use template settings** controls whether the formatting for the table will be controlled by the document properties or the table template.

- If *selected* the table will be formatted from the table template.
- If *cleared* the table template will only control the columns and formatting will use the drawing document properties.

Cut list specific document properties are found in the **Tables > General** category.

Modifying the Cut List Table

Cut list tables can be modified in the same way as Bill of Material and other tables.

- Drag the headers of the table to reorder column and rows.
- Drag the column and row borders to resize.
- Right-click in the table to access modification commands in the shortcut menu. The commands are:

■ **Insert** **Column Right** **Column Left** **Row Above** **Row Below**	■ **Select** **Table** **Column** **Row**
■ **Delete** **Table**	■ **Formatting** **Column Width** **Lock Column Width** **Row Height** **Lock Row Height** **Entire Table**
■ **Sort** **Ascending** **Descending**	■ **Split** **Horizontal Auto Split** **Horizontally Above** **Horizontally Below** **Vertically Left** **Vertically Right**
■ **Merge**	**Save as Template**

29 Add a column to the table.
Right-click in the column labeled DESCRIPTION and select **Insert, Column Right**.

In the PropertyManager, click **Cut list item property** and select **MATERIAL** from the list.

Click **OK** ✔.

30 Add a second column to the table.
Right-click in the column labeled DESCRIPTION and select **Insert, Column Left**.

In the PropertyManager, select **Cut list item name**.

In the **Title** field edit the text if desired and click **OK** ✔.

31 Resize columns and rows as desired.

ITEM NO.	QTY.	CUT LIST ITEM NAME	DESCRIPTION	MATERIAL	LENGTH
1	2	SIDE TUBES	80 x 80 x 6.3	Plain Carbon Steel	750
2	2	FRONT-REAR TUBES	80 x 80 x 6.3	Plain Carbon Steel	740
3	4	LEGS	80 x 80 x 6.3	Plain Carbon Steel	400
4	4	ANGLED BRACES	80 x 80 x 6.3	Plain Carbon Steel	306.274
5	4	BASE PLATES	PLATE, 20 x 120 x 160	Plain Carbon Steel	
6	4	GUSSETS	PLATE, 10 x 125 x 125	Plain Carbon Steel	
7	4	END CAPS	PLATE, 5 x 73.7 x 73.7	Plain Carbon Steel	
8	2	RAILS	75x75x8	AISI 304	900

Tip

Multiple columns or rows can be selected using the **Ctrl** key and selecting the table headers. To change the formatting of all column or rows at once, right-click and select **Formatting, Entire Table**.

The table shown above is formatted as follows:

Column Width of **A**, **B** and **F** = **20mm**

Column Width of **D**, **C**, and **E** = **50mm**

Row Height for Entire Table = **7.5mm**

32 Add balloons.

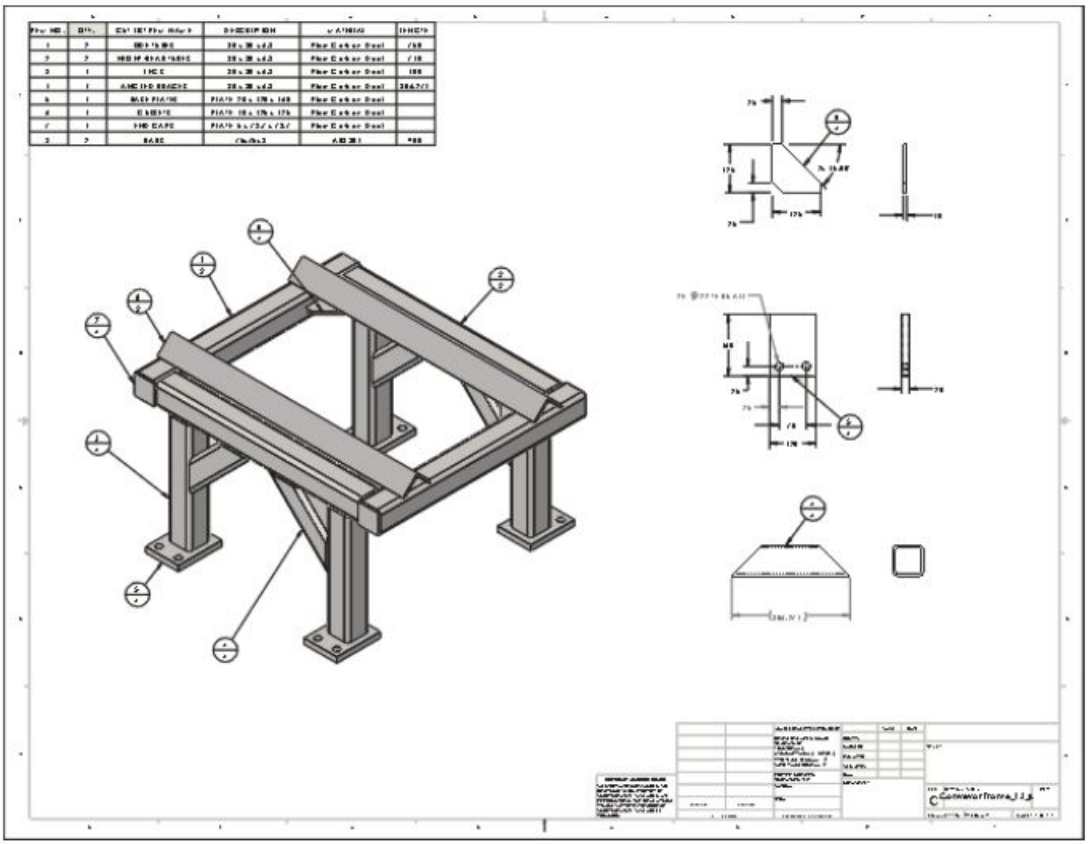
Use **Balloon** or **Auto Balloon** on the Annotations toolbar to create balloon annotations and position as desired.

Use the following settings to create the balloons as shown.

Balloon Settings: Circular Split Line, 2 Characters

Balloon Text: Item Number

Lower Text: Quantity



33 Rename drawing sheet.

Rename Sheet1 to Cut List.

34 Add Sheets.

Add 2 new sheets to the drawing.

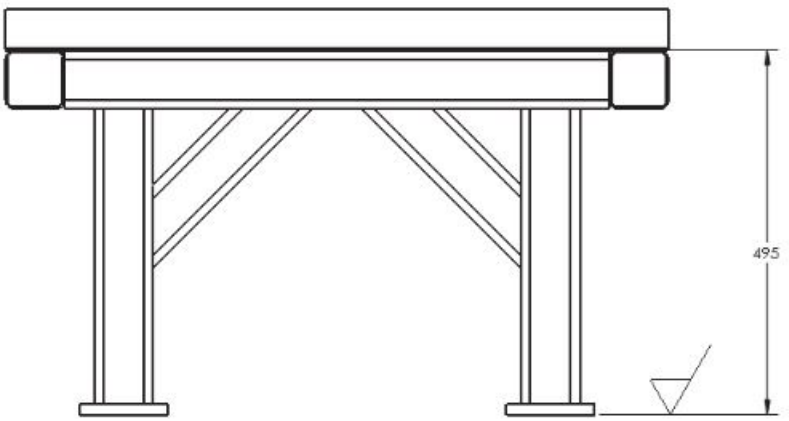
Rename Sheet2 to Weld Detail.

Rename Sheet3 to Machining.

35 Add drawing view.
On the Machining sheet, add a **Front** drawing view. Use Default<As Machined> as the **Referenced Configuration**.

Insert the model item dimension from Cut-Extrude1 or add it using the dimension command.

Add a **Surface Finish** symbol to the bottom face to indicate it will be machined.



36 Save the drawing.

Representing Welds

SOLIDWORKS provides several choices for representing welds:

- **Weld symbols**
  Annotations that can be added to drawing views, parts, or assemblies.
- **Fillet Bead features (page 123)**
  Solid body features added to a multibody part.
- **Weld Bead features (page 126)**
  Graphical features added to a part or assembly.

These techniques can be combined or used individually.

Weld Symbols

Commonly, welds are represented using only weld callout annotations. These annotations can be added directly to a drawing view, or added to the 3D model and imported using the Model Items command.

SOLIDWORKS provides an extensive dialog for defining weld symbols.

Where to Find It

- CommandManager: **Annotations > Weld Symbol**
- Menu: **Insert, Annotations, Weld Symbol**

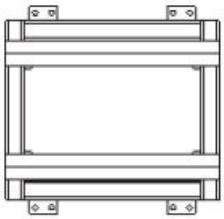
1 Create Standard 3 Views.

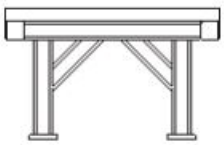
Activate the Weld Detail sheet.

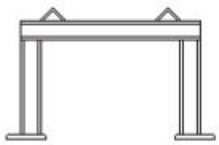
Select the View Layout tab in the CommandManager.

Click **Standard 3 View**.

The **Reference Configuration** will be Default<As Welded>.

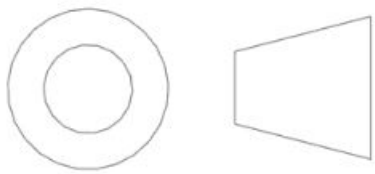






Note

The sheet properties of this template use **Third Angle Projection**.



2 Add Weld Symbol.

Click **Weld Symbol**.

Note

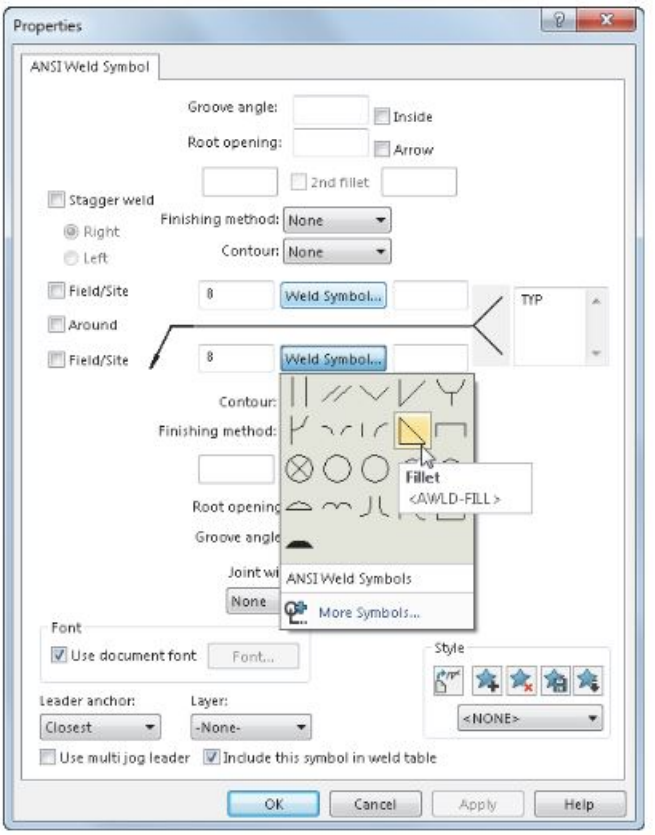
When working in the **Weld Symbol** dialog, move the cursor onto the sheet to see a preview of the symbol. **Zoom** and **Pan** can be used on the sheet while the dialog is open and the dialog can be moved to assist in placing the symbol. The dialog can be left open to add multiple symbols.

3 Fillet weld, both sides.

Click the **Weld symbol** button and select **Fillet** for both sides.

In the dialog, type a size of **8** in the field to the left of the **Weld Symbol** button for both sides.

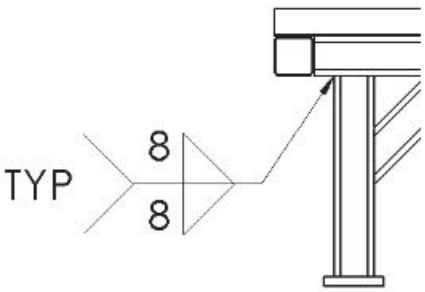
Add a **TYP** note to the symbol.



4 Place weld symbol.

Place the symbol as shown in the **Front** view.

Click **OK** to close the dialog.



5 Flare Bevel with Fillet weld symbol.

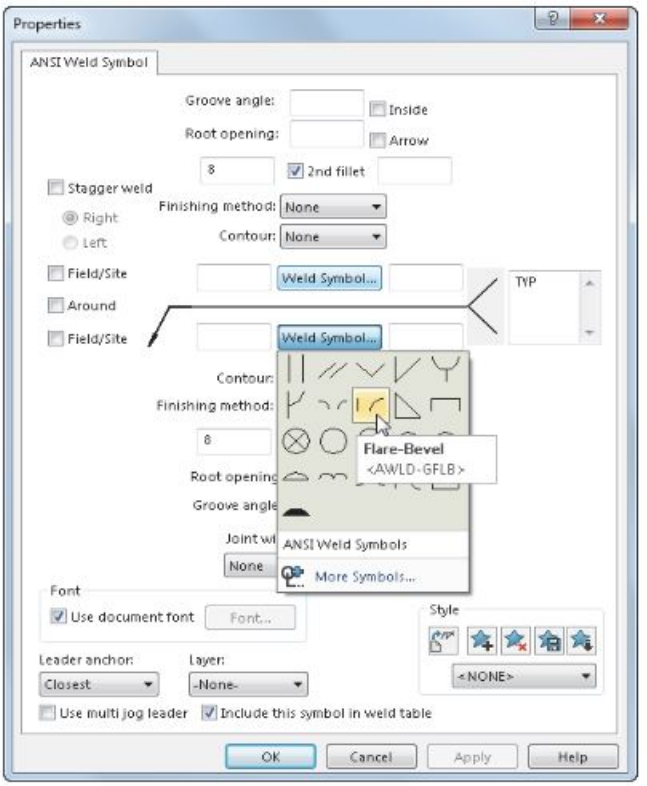
Click **Weld Symbol** to add a new weld symbol.

Click **Weld symbol**, and select the **Flare-Bevel.**

Check the **2nd fillet** option. Use the field to the left of the checkbox to add the size for the fillet as **8mm**.

Add the same callout to the other side of the weld symbol.

Add a **TYP** note to the symbol.

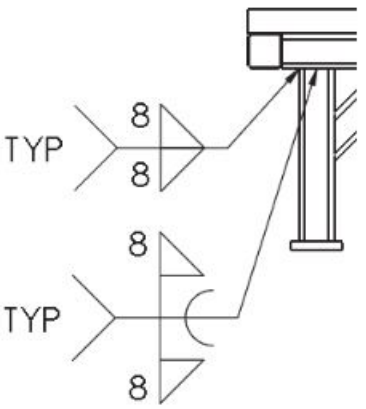


6 Place weld symbol.

Add the symbol to the **Front** view as shown.

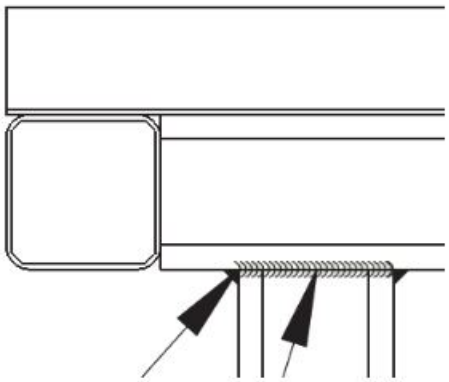
Click **OK** to close the weld symbol dialog.

7 Save the drawing.



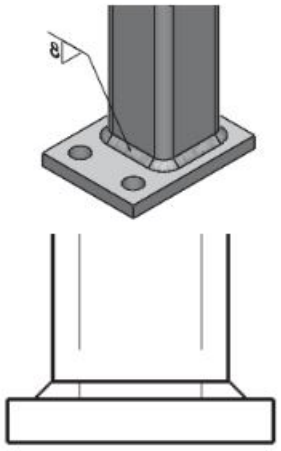
Other Weld Annotations

Besides weld symbols, drawing documents include additional annotations that can be used to represent welds. Under **Insert, Annotations**, add **Caterpillar** or **End Treatments** to edges in a drawing view to visually display a weld bead.



Fillet Beads

The **Fillet Bead** feature creates solid bodies to represent welds on a model. As a solid feature, fillet beads can be manipulated in the same was as other solid geometry: they can be patterned and cut, they are included in mass calculations, and they are fully represented in drawing views. However, as a solid feature, many fillet beads may affect the performance of a model. Also, only *fillet* weld types are available and they can only be applied to weldment part models.



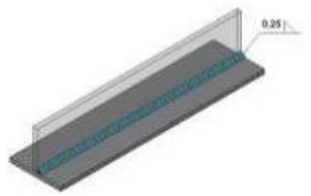
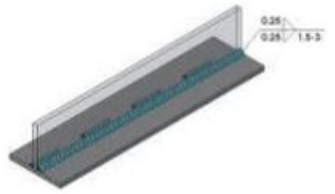
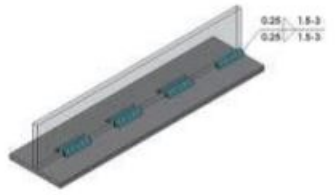
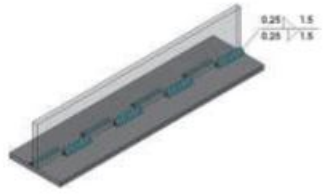
Fillet Beads are typically used when required for interference detection and analysis operations.

Fillet Beads are created by selecting faces from 2 separate bodies. There can be a gap between selected faces or bodies as long as the gap is smaller than the specified weld bead size.

Options exist for a full length or intermittent weld bead, one side or both sides. It is possible to combine full length and intermittent weld beads on opposite sides.When creating intermittent beads on both sides, they can be staggered if desired.

If weld symbols are created with the Fillet Bead feature, they can be imported into drawing views using the **Model Items** command.

Some examples of fillet weld beads are shown in the following table. Note that the vertical plate has been made transparent for illustration purposes.

Arrow Side: Full Length **Other Side: None**	
Arrow Side: Full Length **Other Side: Intermittent**	
Arrow Side: Intermittent **Other Side: Intermittent**	
Arrow Side: Staggered **Other Side: Staggered**	

All weld features can be suppressed in the part by right-clicking the Weldment feature and selecting **Suppress Weld Beads**.

Where to Find It

- Menu: **Insert, Weldments, Fillet Bead**

Tip

The Weldments tab of the CommandManager can be customized to add **Fillet Bead** if used frequently.

---

8 **Open the Conveyor Frame part.**
Activate the Conveyor Frame part document window or open it from a drawing view.

9 **Activate the Default<As Machined> configuration.**
Working in the top level configuration will ensure the weld features will exist in both configurations.

10 Add a **Fillet Bead.**

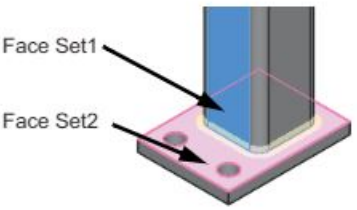
Click **Fillet Bead** .

Under **Arrow Side**, select the following:

- **Bead type** = **Full length**
- **Fillet size** = **8mm**
- **Tangent propagation**: selected

11 **Select faces.**

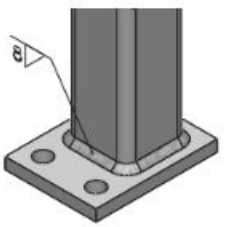
For **Face Set1** select the front, flat face of the leg. For **Face Set2** select the upper face of the Plate.



Click **OK** .

The results are shown at the right.

The weld symbol will only be visible if the **All Annotations** choice turned on in the **View** menu.



12 **Reorder.**

Drag the Fillet Bead1 feature so it is before the Mirror1 feature in the FeatureManager design tree.

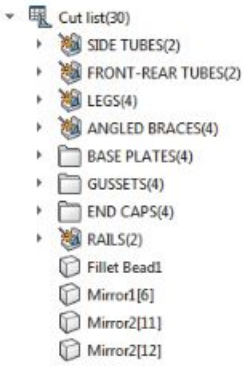
13 **Edit the Mirror features.**

Edit the definition of both Mirror features and include the fillet weld bead in the **Bodies to Mirror** list.

14 **Results.**

Evaluate the **Cut List** folder.

Bodies associated with the **Fillet Bead** are not grouped into cut list items and will not appear in a cut list table.



15 **Modify Display State.**

Activate the Base Plate Detail display state.

Hide these new bodies in this display state.

Activate <Default>_Display State1.

16 **Save the part.**

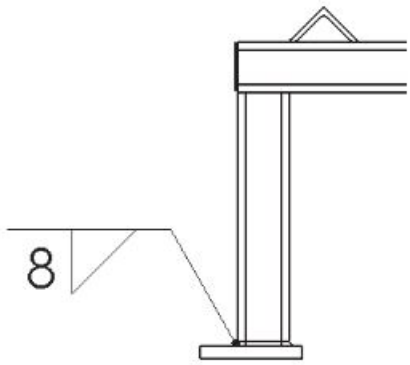
17 Drawing view annotations.
Activate the drawing document window.

Click **Model Items**.

Source: Selected feature

Annotations: Weld Symbols

In the **Right** View, select the **Fillet Bead**.



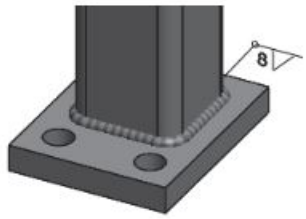
Click **OK** ✔.

18 Modify weld symbol.
Existing weld symbols can be modified by double-clicking.

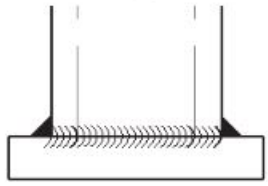
Edit the fillet weld symbol to add a **TYP** note.

Weld Bead Feature

Weld Beads are graphical features added to a 3D model to represent welds. Weld Beads do not create any solid geometry, so therefore are very lightweight and do not have much affect model performance.



When detailing, **Model Items** can be used to add weld beads as **Caterpillar** or **End Treatment** annotations or display associated weld symbol in drawing views.



Weld Beads automatically include property information such as length, and can evaluate values such as cost (see *Weld Bead Properties* on page 133). This property information can then be displayed in Weld Tables. **Weld Beads** are most useful for capturing this type of property information as well as for cosmetically displaying welds on a model. **Weld Beads** features are not limited to use in weldment models; they can also be used in assemblies and standard parts.

Key features of weld beads:

- Compatibility with all types of geometry and weld types.
- Inclusion of weld bead properties for use in drawings using weld tables.
- Automatically created weld symbols.
- Association of weld symbols with the weld beads.
- A **Weld Folder** contains all the weld beads in the FeatureManager tree.

All weld features in the part can be suppressed by right-clicking the Weldment feature and selecting **Suppress Weld Beads**.

Where to Find It

- CommandManager: **Weldments > Weld Bead**
- Menu: **Insert, Weldments, Weld Bead**

---

1 **Open the Conveyor Frame part.**
Activate the Conveyor Frame part document window or open it from a drawing view.

2 **Activate the Default<As Machined> configuration.**
Working in the top level configuration will ensure the weld features will exist in both configurations.

3 **Weld bead.**
Click **Weld Bead**.

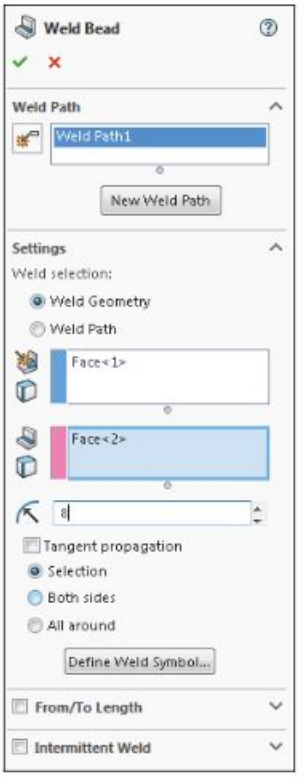
The first **Weld Path** will be Weld Path 1.

Weld Bead Options

Multiple **Weld Paths** can be created within the same **Weld Bead** PropertyManager. A weld path is defined by either the virtual intersection of selected faces if using the **Weld Geometry** option, or by selecting edges or sketches if using the **Weld Path** option.

When selecting **Weld Geometry**, the **Smart Weld Selection Tool** can be used to quickly select faces. With the **Smart Selection** tool active, drag the cursor over faces to make selections.

The options in **Settings** apply per weld path. Once the settings have been defined for the current weld path, click **New Weld Path** to define the next weld bead. Existing paths can be selected in the **Weld Path** group box to modify settings at any time.

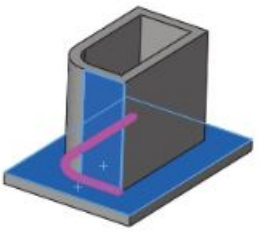


Tip

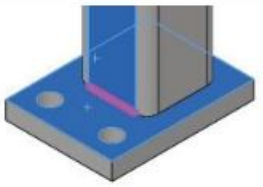
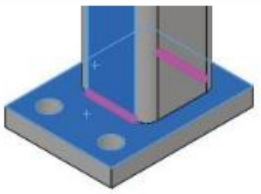
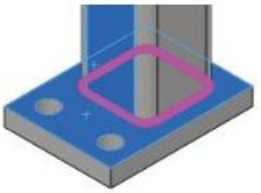
The shortcut menu provides the option for **New Weld Path** when creating **Weld Beads** features.

Settings for each weld bead path may vary based on selections. The settings and options that can be found are as follows:

- **Weld selection** - Lists the faces, edges, or sketches where the weld bead is applied.
- **Bead size** - Sets the bead thickness.
- **Tangent propagation** - Applies the weld bead to all edges that are tangent to the selected face or edge.

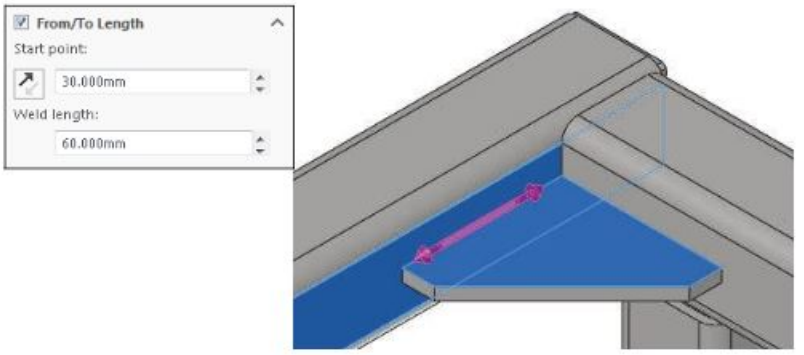


With **Tangent propagation** cleared, options exist for:.

Selection	
Both Sides	
All around	

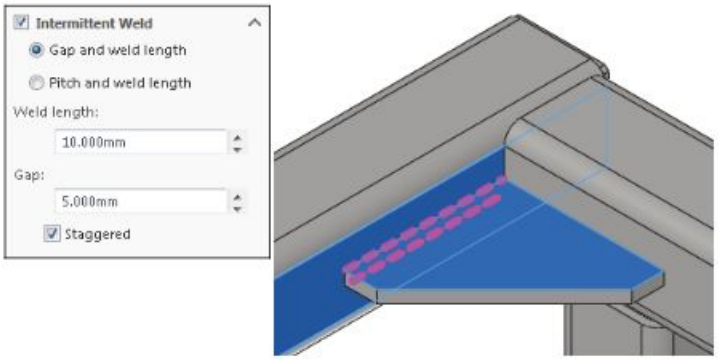
Define Weld Symbol - launches dialog for customizing the weld symbol associated with the weld bead. The default symbol is a fillet weld.

From/To Length - Use the settings below or the handles in the graphics area to manipulate where the weld bead begins and ends.



- **Start point** - Defines where to start the weld bead from the first end.
- **Reverse Direction** - Starts the weld bead from the opposite end.
- **Weld length** - Sets the length of the weld bead.

Intermittent Weld - Use the settings below to define an intermittent weld bead.



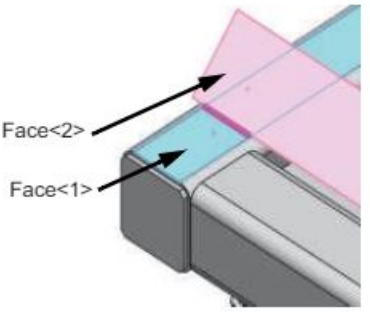
- **Gap and weld length** - Allows the intermittent weld to be defined by specifying the length of each weld and the gap distance between them.
- **Pitch and weld length** - Allows the intermittent weld to be defined by specifying the length of each weld and the pitch (the distance from one weld to the next).
- **Staggered** - If intermittent welds are being added to **Both sides**, they can be staggered rather than in-line with each other.

4 **Select faces.**
Under **Settings, Weld selection:** use the **Weld Geometry** option.

In the Weld From selection box pick the top face of the Side Tube.

In the Weld To selection box pick the face of the Rail as shown.

The weld path preview shows as magenta.



Tip

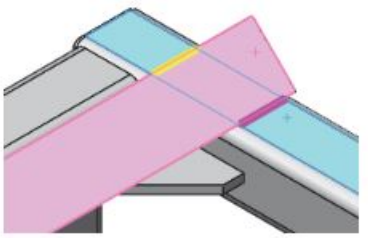
When the cursor displays a mouse with a blue arrow over the right button use the right mouse button to activate the next selection box in the PropertyManager.

5 **Weld Bead Settings.**
Use the following settings:

- **Fillet size = 8mm**
- **Selection**

6 **New Weld Path.**
Click **New Weld Path**.

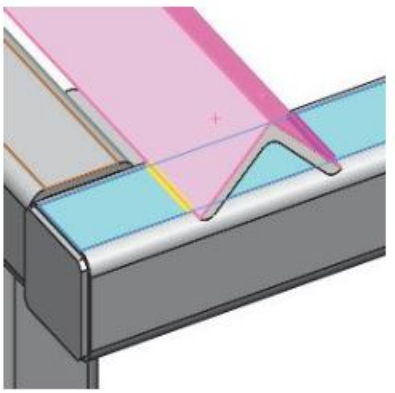
Make selections to add a weld path to the opposite side of the Rail.



7 **New Weld Path with multiple faces.**
Click **New Weld Path**.

Select the top face of the opposite Side Tube and 2 faces of the Rail as shown.

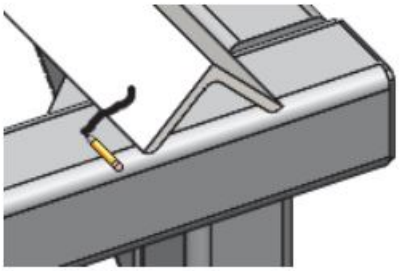
As long as the selections belong to the same bodies, multiple weld paths can be automated in this way.



8 Smart Weld Selection Tool.
Click **New Weld Path**.

Click the **Smart Weld Selection Tool**.

Drag the cursor over the faces of the back Rail and Side Tube.



Click **New Weld Path** and repeat to complete the remaining weld paths for the back Rail.

Tip

The shortcut menu provides an option for **New Weld Path** when working in this command. To access the shortcut menu, move the cursor slightly to cancel the **OK** shortcut which appears at the cursor after a weld path creation, then right-click.

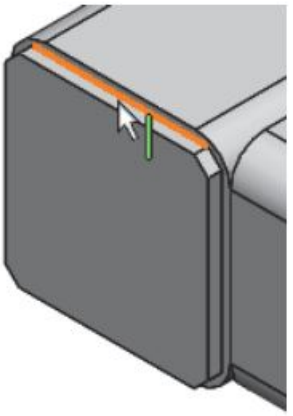
9 Select using Weld Path.
If an existing edge or sketch would correctly represent the weld path, the **Weld Path** selection option can be used.

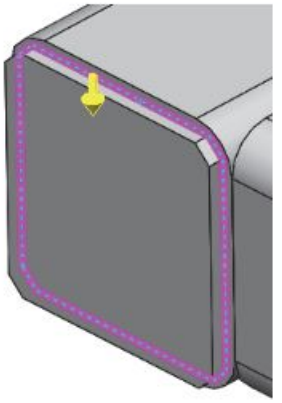
Turn *off* the **Smart Weld Selection Tool** and click **New Weld Path**.

Under **Settings**, click **Weld Path**.

Right-click an edge of the End Cap as shown and click **Select Loop**.

Use the arrow to change which connected loop is being selected, if necessary.





Change the bead size to **3mm**.

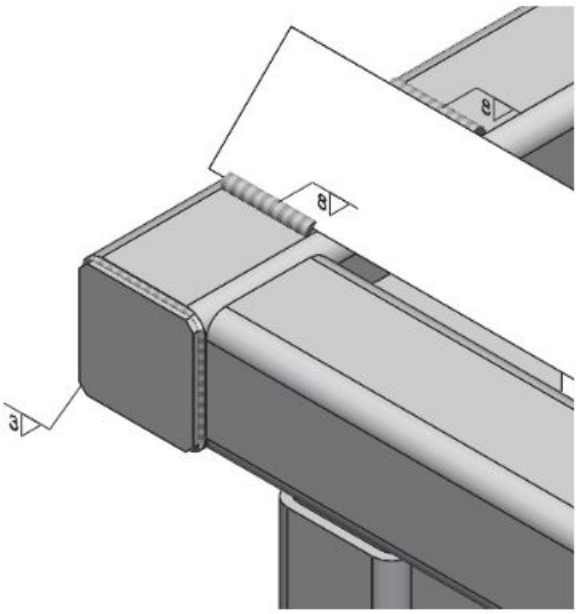
10 New Weld Paths.
Select new weld paths for each remaining End Cap.

Click **OK** ✔.

11 Results.

The result is shown below.

To see both the weld symbols and the weld beads, the **Hide/Show** options must be turned on for **All Annotations** and **Weld Beads**.



Tip

To turn only weld symbol annotations (specifically) off, go to **Tools, Options, Document Properties**, **Detailing**, under **Display filter**, and clear the **Welds** check box.

Weld Folder

Weld Beads are stored in a folder near the top of the FeatureManager tree. Within the folder, weld beads of the same size are grouped together into sub folders. Each weld subfolder holds properties associated with the welds, similar to cut list item folders.

Individual weld paths can be accessed within each subfolder and list the length of each path. A weld path can be edited by right-clicking the path in the folder or graphics area and clicking **Edit Feature**.

All Weld Beads can be hidden by right-clicking the Weld Bead folder and clicking **Hide Cosmetic Welds**.

Weld Bead Properties

Use the Weld Bead Properties dialog box to view and set weld bead properties. These properties can be used in Weld Tables in drawings.

The Weld Bead Properties dialog box consists of:

Weld Folders	The left column of the dialog box lists each unique group (same type and size) of weld beads. When an item is selected in this column, any changes made in the Properties column apply to all weld beads in the group.
Properties	Weld Material
	Weld Process
	Weld Mass per Unit Length
	Welding Cost per Unit Mass
	Welding Time per Unit Length
	Number of weld passes
Evaluated Values	Total Number of welds
	Total Weld Length
	Total Weld Mass
	Total Welding Cost
	Total Welding Time

Where to Find It

- Shortcut menu: Expand the Weld Folder, then right-click a Weld subfolder and click **Properties**

12 Add weld properties.

Expand the **Weld Folder**.

Right-click 8mm Fillet Weld and click **Properties**.

Select the **Weld Material** and type 309L SS.

Select the **Weld Process** and type GTAW.

13 Properties to additional welds.

While still in the Weld Bead Properties dialog, click 3mm Fillet Weld.

Select the **Weld Material** and type CARBON STEEL.

Select the **Weld Process** and type SMAW.

Click **OK**.

Weld Bead Information on Drawings

When weld bead features and annotations exist in a model, they can be added to drawing views using the **Model Items** command.

Weld Bead features can import the following annotations into drawing views:

- **Weld Symbols**
- **Caterpillars**
- **End Treatments**

The weld property information can also be displayed in a drawing through the use of a **Weld Table**.

---

14 Weld Bead drawing annotations.

Activate the drawing document window.

Click **Model Items** and use the following options:

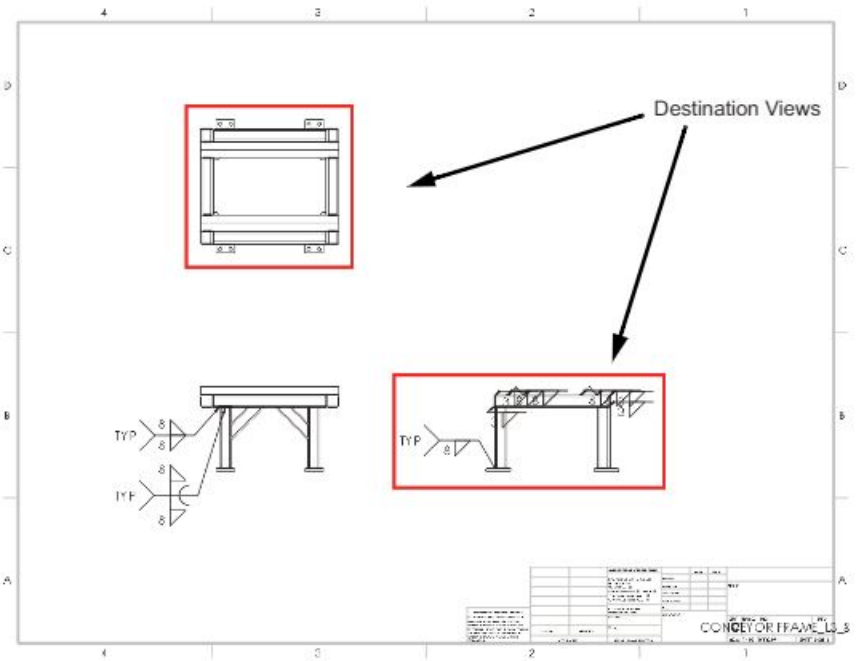
Source: Entire model

Destination view(s): Top View & Right View

Dimensions: *Remove* all selections

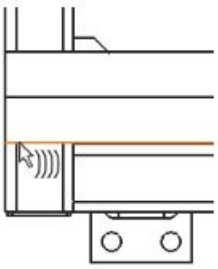
Annotations: Weld Symbols, Caterpillars, End Treatments

Click **OK** .



Tip

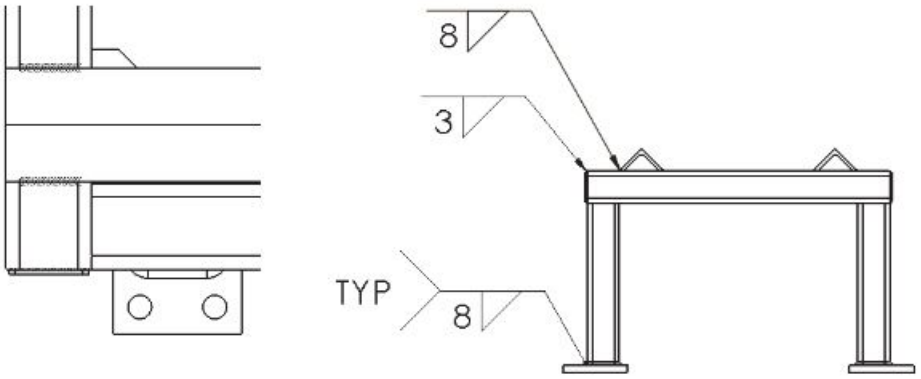
If adding Model Items for only selected weld bead features, use the cursor feedback and tool tips to select edges with associated weld paths.



15 Results.

Annotations are added to the selected views.

Optional: Unnecessary weld symbols can be deleted or hidden.



Weld Tables

Weld Tables display information about welds in a model. The properties associated with Weld Bead features are imported into the table. Other weld symbols can also be added to the table.

Weld tables can be modified in the same ways as Cut List tables by using the shortcut menu. See *Modifying the Cut List Table* on page 116 for more information.

Where to Find It

- CommandManager: **Annotations > Tables > Weld Table**
- Menu: **Insert, Tables, Weld Table**

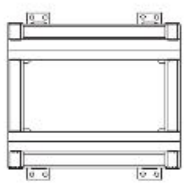
16 Weld table.

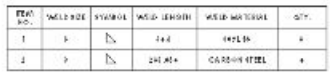
Click **Weld Table** .

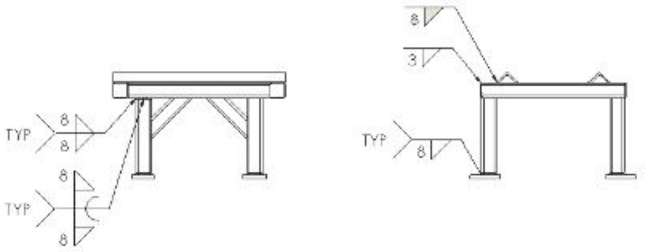
Select a view of the weldment. Click **OK** .

Click in the upper right corner of the drawing to place the table.

Only **Weld Bead** features are included in the table by default.







17 Add drawing annotations to table.

Click the upper left corner of the table to access the **Weld Table** PropertyManager.

Click **Include drawing annotations**.

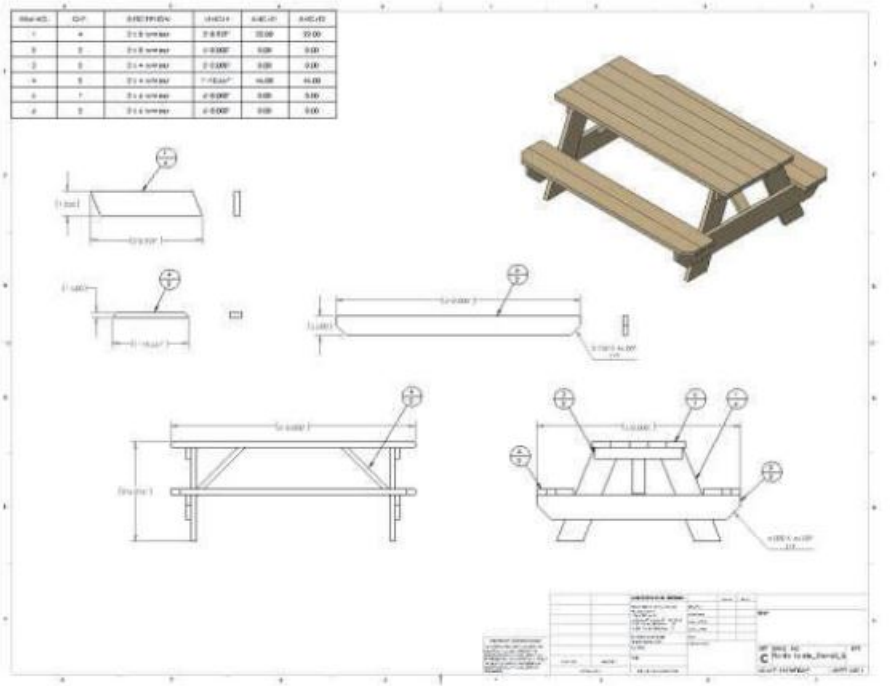
Cells of the table can be modified to adjust values in the table if desired.

ITEM NO.	WELD SIZE	SYMBOL	WELD LENGTH	WELD MATERIAL	QTY.
1	8		54.8	309L SS	8
2	3		283.084	CARBON STEEL	4
3	8				2
4					2
5	8				1

18 Save and close all files.

Exercise 8: Detail Picnic Table

Create a detail drawing of the Picnic Table as shown.



This lab reinforces the following skills:

- *Select Drawing View Bodies* on page 108
- *Using Relative View* on page 113
- *Cut List Tables* on page 115
- *Modifying the Cut List Table* on page 116

Units: **feet & inches**

Procedure

1 Open an existing part.
Use the model completed in *Exercise 6: Picnic Table* on page 94 or open the part provided in the `Lesson03\Exercises` folder, `Picnic Table_Detail`.

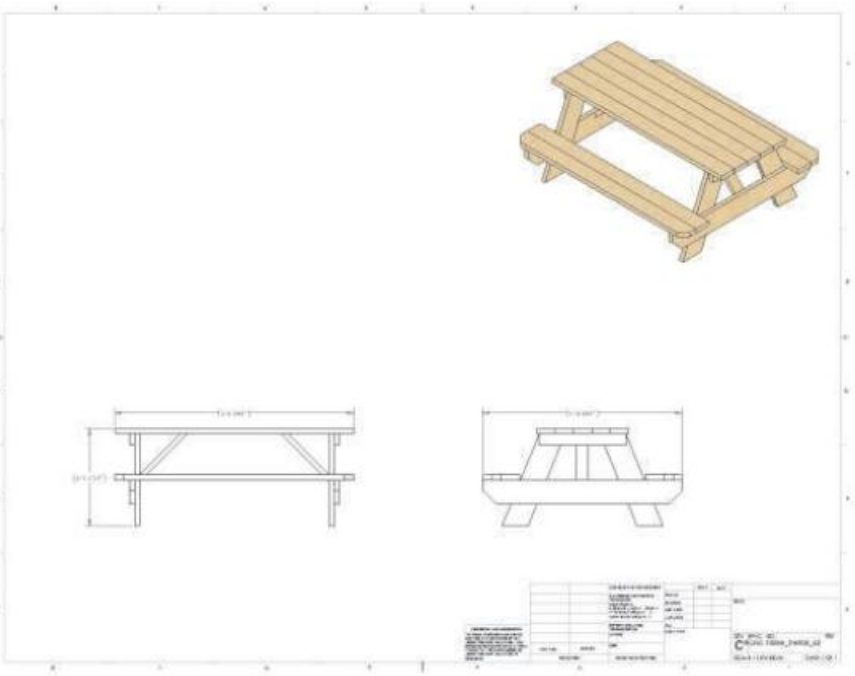
2 Verify model is prepared for detailing.
Review the **Cut List** folder and `Cut List Properties` dialog. Verify that all bodies are part of a cut list item and have appropriate properties.

3 Create drawing from part.
Click **Make Drawing from Part/Assembly** on the Standard toolbar. Use the `C_Size_ANSI_Inch` template.

4 Create a drawing views.
Create front, right, and isometric views as shown.

Change the drawing view properties for the isometric view to be **Shaded with Edges**.

5 Optional: Add dimensions.
Add reference dimensions as shown.



6 Create a view with selected bodies.
Create a **Top** view.

Use the **Drawing View** PropertyManager to **Select Bodies**.

Select the chamfered bench piece.

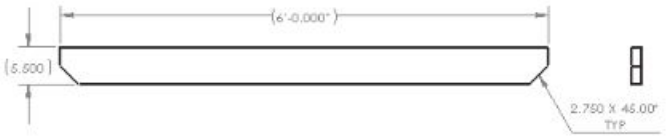
Tip

To prevent drawing views from aligning, hold the **Ctrl** key down when placing the view on the sheet.

7 Projected View.

Create a **Projected View** to the right.

8 Optional: Add dimensions.
Add reference dimensions and a chamfer dimension as shown.



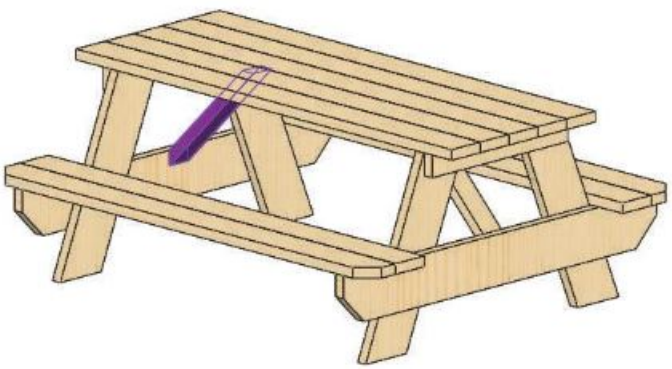
Tip

The **Chamfer Dimension** tool is available in the **Smart Dimensions** flyout menu. It requires 2 selections, the chamfer edge and the edge the angle should be measured from.

9 Create a Relative View.
To detail the angled brace, a view of the individual body will be created. Since none of the default view orientations of the model would align the body properly, use a **Relative View**.

Right-click an existing drawing view. Select **Drawing Views, Relative View** .

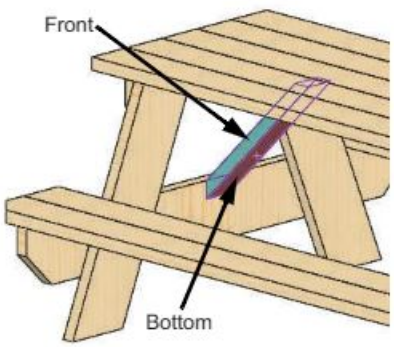
For **Scope**, click **Selected Bodies** and select the angled brace as shown.



10 Orientation.
Make sure **First orientation** is set to **Front** and select the front face of the angled brace.

Set **Second orientation** to **Bottom**. Select the face shown.

Click **OK** .

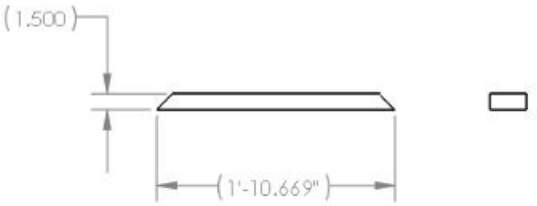


11 Place view.
Click on the drawing sheet to position the view.

12 Projected View.
Create a **Projected View** to the right.

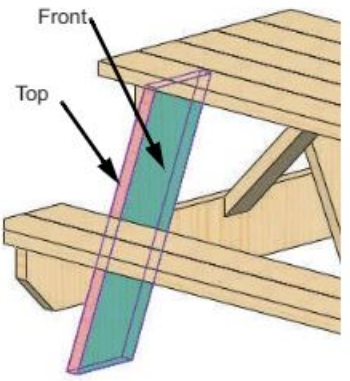
13 Optional: Add dimensions.

Add reference dimensions as shown.



14 Add another Relative View.

Add a **Relative View** using the body and faces as shown.

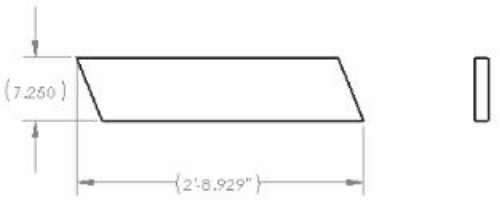


15 Projected View.

Create a **Projected View** to the right.

16 Optional: Add dimensions.

Add reference dimensions as shown.



17 Add a Cut List Table.

Add a **Weldment Cut List** to the upper left of the drawing sheet.

18 Modify table.

Use the shortcut menu to insert 2 new columns to the right as shown. Use the `Angle1` and `Angle2` cut list item properties.

Optional: Format the table as desired.

ITEM NO.	QTY.	DESCRIPTION	LENGTH	ANGLE1	ANGLE2
1	4	2 x 8 Lumber	2'-8.929"	22	22
2	2	2 x 8 Lumber	5'-0.000"	0	0
3	2	2 x 4 Lumber	2'-2.000"	0	0
4	2	2 x 4 Lumber	1'-10.669"	45	45
5	7	2 x 6 Lumber	6'-0.000"	0	0
6	2	2 x 6 Lumber	6'-0.000"	0	0

19 Add balloons.

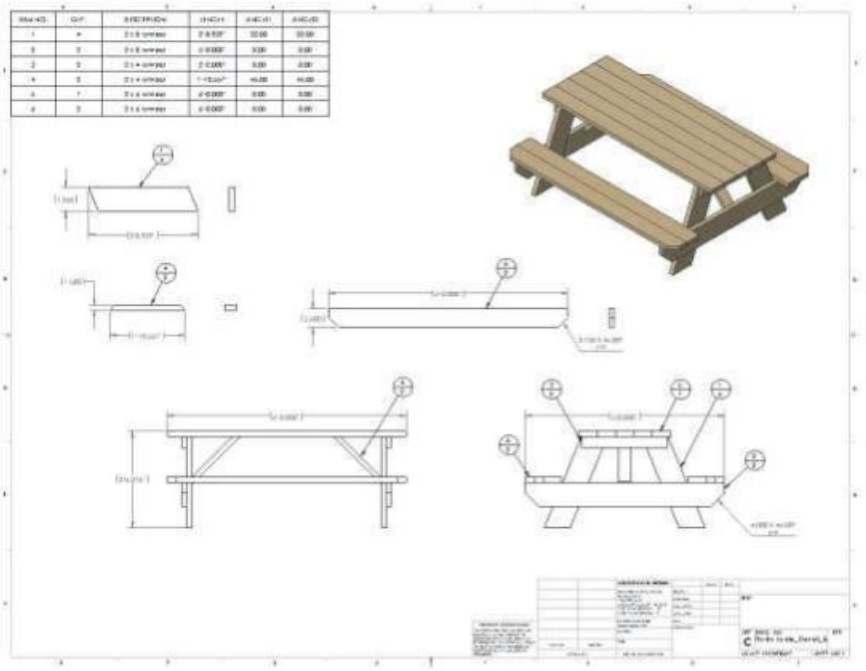
Use **Balloon** or **Auto Balloon** on the Annotations toolbar to create balloon annotations and position as desired.

Use the following settings to create the balloons as shown.

Balloon Settings: Circular Split Line, 2 Characters

Balloon Text: Item Number

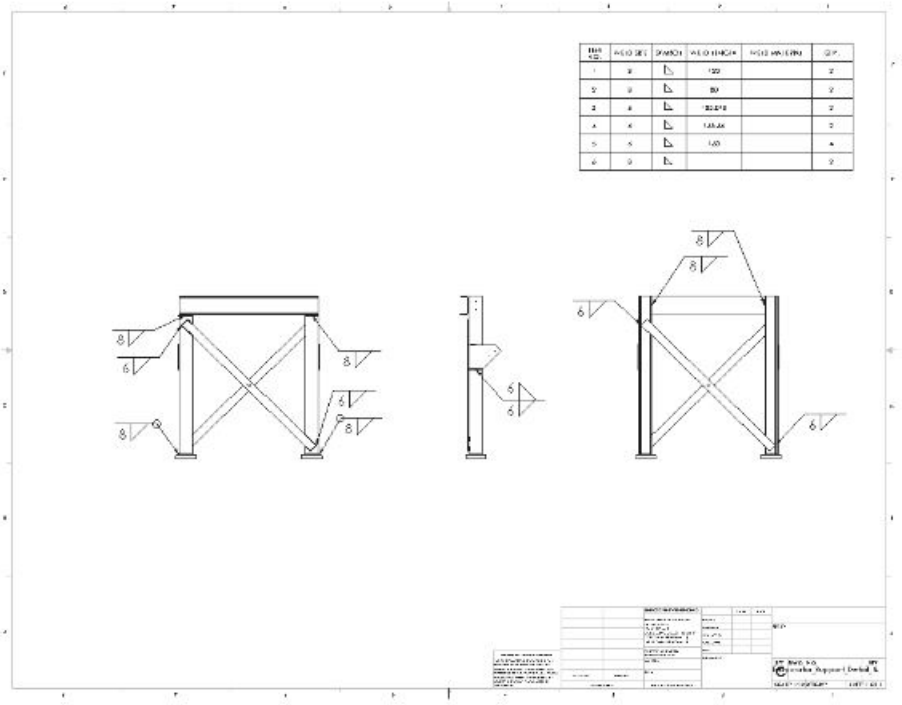
Lower Text: Quantity



20 Save and close all files.

Exercise 9: Representing Welds

Use different techniques to represent welds for the Evaporator Support and create a detail drawing.



This lab reinforces the following skills:

- *Feature Scope* on page 106
- *Using Display States* on page 110
- *Isolate* on page 110
- *Fillet Beads* on page 123
- *Weld Bead Feature* on page 126
- *Weld Tables* on page 135

Procedure

1 Open an existing part.

Use the model completed in *Exercise 4: Evaporator Support* on page 55 or open the part provided in the Lesson03\Exercises folder, Evaporator_Support_Detail.

2 Bolt holes.

Some of the connections in the frame will be bolted.

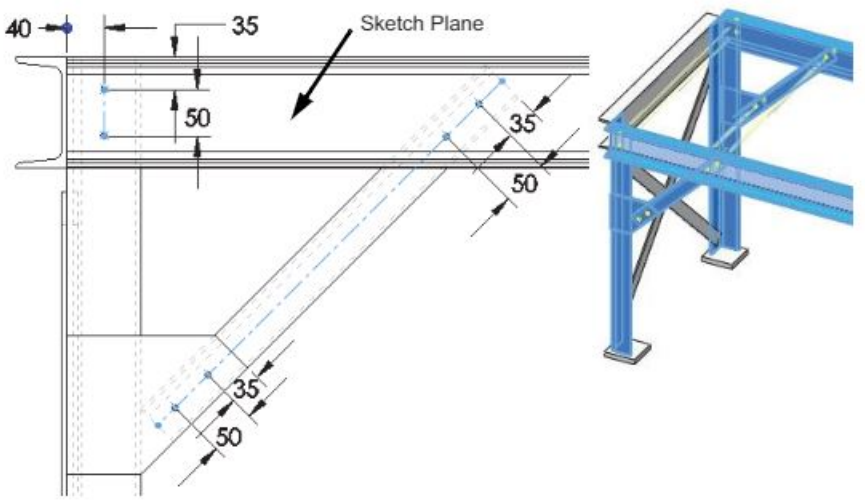
Use the **Hole Wizard** to add clearance holes for **6mm** bolts as shown in the illustration below. Use the end condition **Through All**.

Tip

Hole Wizard features only go in one direction, so be sure to sketch on the outermost face available or create a plane in the appropriate position.

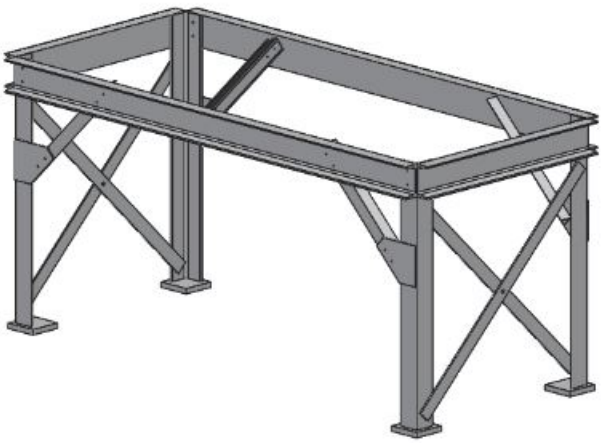
Use the **Auto select** option in **Feature Scope** to ensure the holes affect all visible bodies in the model. This is the default setting.

In production, these holes will be added prior to welding, so there is no need to configure the feature.



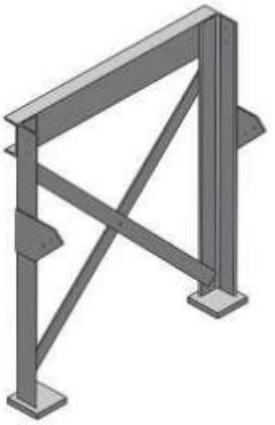
3 Mirror the bolt holes.

Mirror the bolt hole feature with respect to the Right Plane.



4 Create a Display State.

Use **Ctrl** to select faces from each of the bodies shown and click **Isolate.**



These pieces require welding.

Click **Save as Display State** from the Isolate dialog. Name the display state Welded End.

Welds will be added to this area of the model and then detailed in a drawing.

5 Fillet Bead.

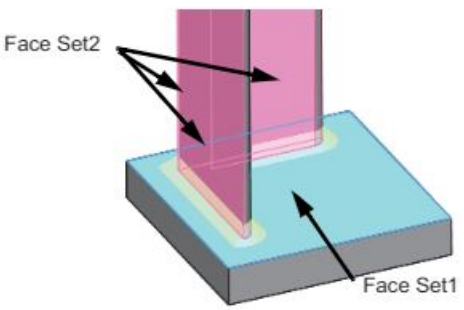
Add a **Fillet Bead** feature to represent the weld at one of the base plates. Use the following settings:

- **Bead type** = **Full length**
- **Fillet size** = **8mm**
- **Tangent propagation**: selected

6 Select faces.

For **Face Set1** select the top face of the base plate.

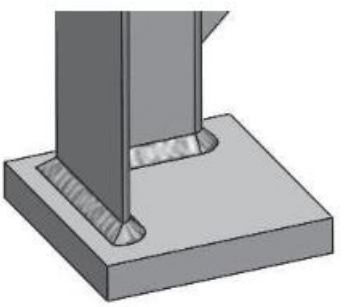
For **Face Set2** a minimum of 3 faces will need to be selected from the angle to allow the fillet weld to be all around.



Clear the option to **Add weld symbol**. The proper symbol will be added manually in the drawing.

Click **OK** ✔.

The results are shown at the right.



7 Reorder and Mirror.

Reorder the Fillet Bead1 feature so it precedes the Mirror1 feature in the tree.

Edit Mirror1 to include the fillet bead solid body.

8 Weld Bead.

Click **Weld Bead** .

For `Weld Path1` use the following settings:

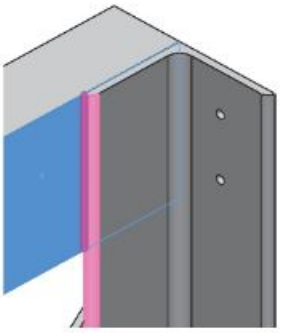
Weld Selection: Weld Geometry

Weld From: Face of top beam

Weld To: Face of leg

Bead Size: 8mm

Selection



9 New Weld Path.

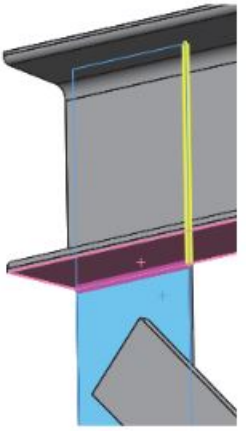
Click **New Weld Path**.

Add the same weld bead to the other side.

10 New Weld Path.

Click **New Weld Path**.

Create additional **8mm** weld beads between legs and top beam as shown.



11 Weld paths for cross braces.

Add new weld paths for each end of the cross brace bodies using the following settings:

Weld Selection: Weld Geometry

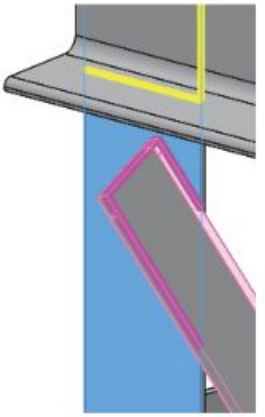
Weld From: Face of leg

Weld To: 3 faces of cross brace

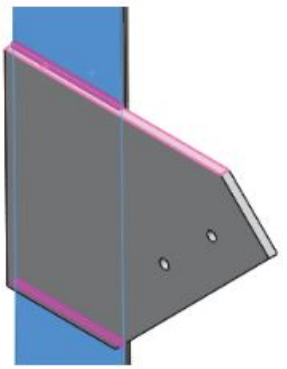
Bead Size: 6mm

Selection

Click **OK** ✔ .



12 Weld Paths for reinforcing plates.



Click **Weld Bead** .

Add a weld to both sides of the reinforcing plate by using the following settings:

Weld Selection: Weld Geometry

Weld From: Face of leg

Weld To: Face of reinforcing plate

Bead Size: 6mm

Both Sides

Create a new weld path of the same type for the other reinforcing plate.

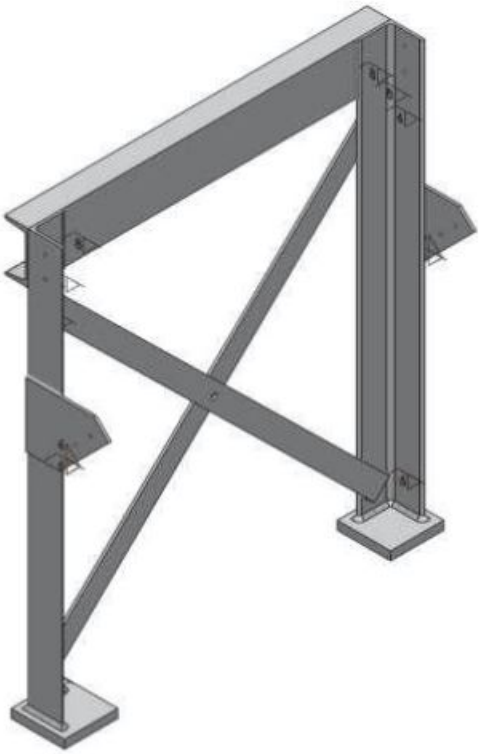
Click **OK** ✔ .

Tip

The automatic weld symbols work better if different types of welds are added as separate features rather than new weld paths.

13 Results.

The result is shown at the right.



To see both the weld symbols and the weld beads, the **View** options must be turned on for **All Annotations** and **Weld Beads** .

14 Exit Isolate.

Click **Exit Isolate.**

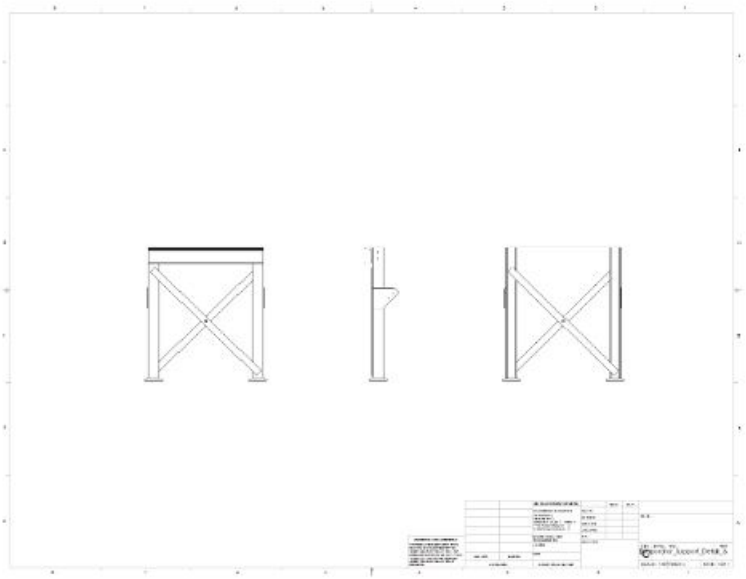
Save the part.

15 Create a drawing.

Create a new drawing using the C_Size_ANSI_MM template from the Training Templates.

Create a **Front** view. Use the Default<As Machined> configuration and the Welded End display state.

Create **Projected Views** to the right and left. Reposition the views as desired.

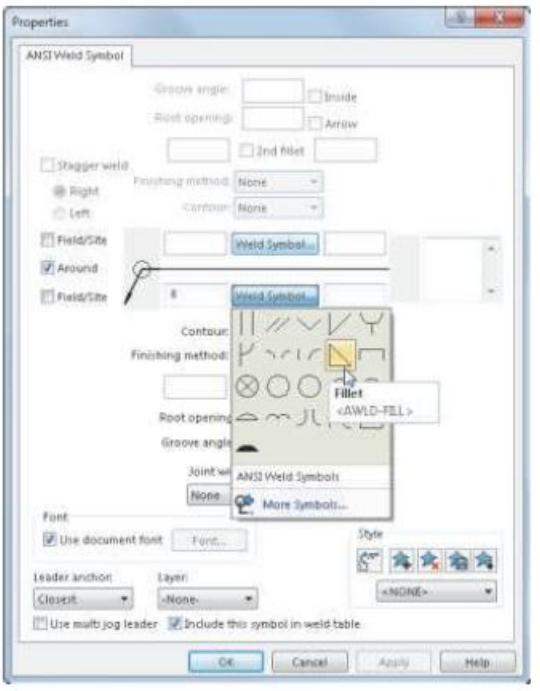


16 Weld Symbol.

To add the weld symbols for the fillet bead features, click **Weld Symbol**.

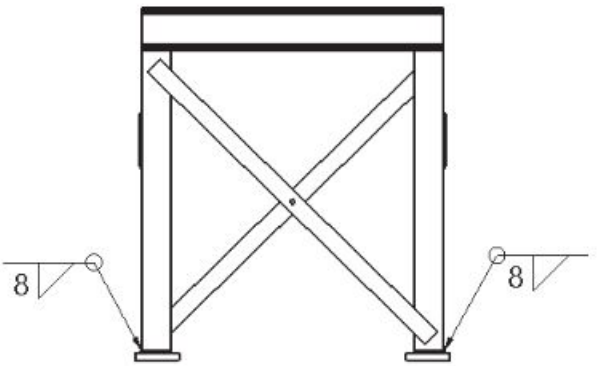
In the weld dialog, click **Weld Symbol** and select the **Fillet**.

Type **8mm** for the size and check **Around.**



17 Place weld symbol.

Click an edge of the Fillet Bead feature to place the arrow head. Click again on the sheet to place the symbol. Add a second instance of this weld symbol to the other fillet bead.



Tip

Click multiple locations in the drawing while the dialog is open to create multiple instances of a weld symbol.

Click **OK** to close the dialog.

18 Model Items.

To add the annotations associated with the weld bead features, model items will be added.

Click **Model Items**.

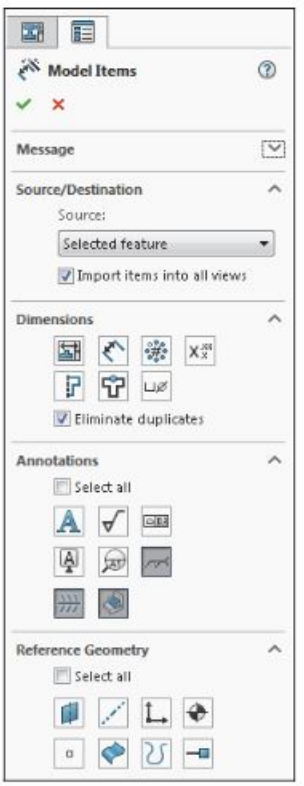
For **Source**, choose **Selected feature** and **Import into all views.**

Selecting features allows for greater control over where the weld symbol annotations will be added.

For **Dimensions**, **clear** all options.

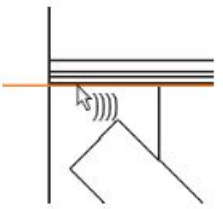
No dimensions are needed for this weld detail drawing.

For **Annotations**, select **Weld Symbols**, **Caterpillars**, **End Treatments**.



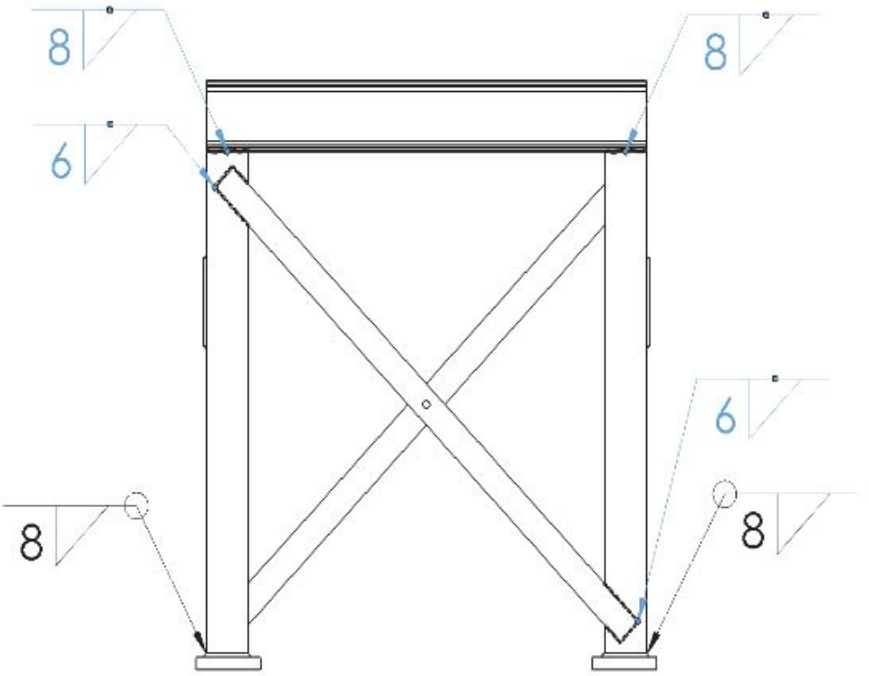
19 Select features.

Use tool tips and cursor feedback to assist in selecting weld bead annotations.

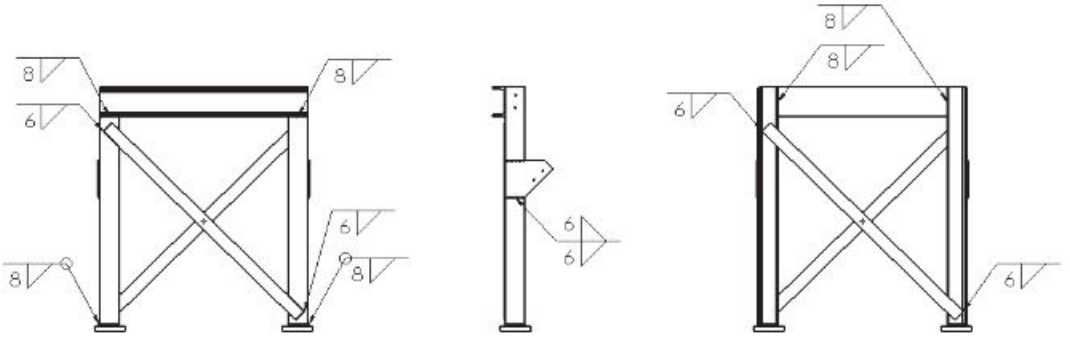


Select the 4 weld bead features in the left view to generate the annotations as shown.

Reposition symbols as desired.



Select features in the remaining 2 views to create annotations as shown.



20 Add weld table.
Click **Weld Table** .

Select a view of the weldment.

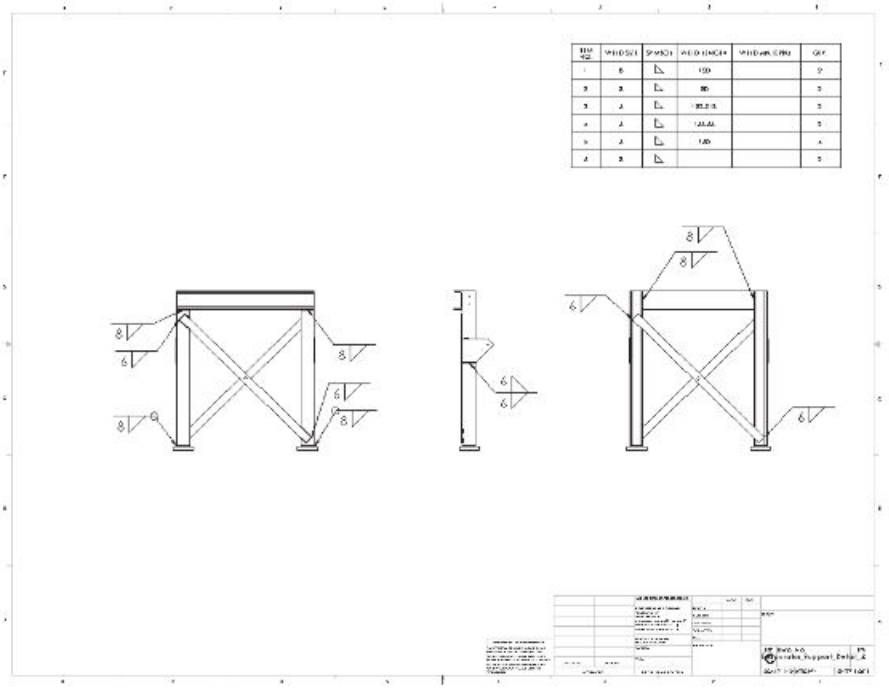
To include the symbols manually added to the drawing, select **Include drawing annotations** under **Options**.

Click **OK** .

Click in the upper right corner of the drawing to place the table.

ITEM NO.	WELD SIZE	SYMBOL	WELD LENGTH	WELD MATERIAL	QTY.
1	8		120		2
2	8		80		2
3	6		183.018		2
4	6		165.66		2
5	6		160		4
6	8				2

21 Results.



22 Save and close all files.

Lesson 4
Working with Bent Structural Members

Upon successful completion of this lesson, you will be able to:

- Create 3D sketches.
- Create 3D sketch planes.
- Create bent structural members.

Working with Bent Structural Members

When working with weldments, options exist to merge sketch segments together to represent bent members. Often pieces are bent in more the 2 directions, so 3D sketching is a common requirement.

In the frame of the Hand Truck pictured, the perimeter will be produced as a single bent tube. To accomplish this, the layout for this member will created in a 3D Sketch.

3D Sketching

As the name implies, the entities in a 3D sketch are not constrained to a single plane as they are in a traditional 2D sketch. This makes 3D sketches very useful for applications such as sweeping, lofting, and producing bent structural members. However, free-hand 3D sketching can sometimes be a bit of a challenge. To be successful, understanding the on-screen feedback and relations in the 3D sketching environment is key.

Where to Find It

- CommandManager: **Weldments > 3D Sketch**
- CommandManager: **Sketch > Sketch Flyout > 3D Sketch**
- Menu: **Insert, 3D Sketch**

Using Reference Planes

3D sketching with standard reference planes allows sketch entities to be created in 3D by switching between planes that exist in the model. To switch between the default planes of the model, press **Tab** while the sketch tool is active.

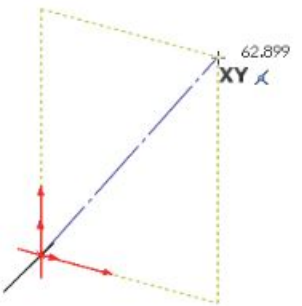
Feedback from the cursor will indicate which standard plane you are sketching on by displaying:

- **XY** = Front Plane
- **YZ** = Right Plane
- **XZ** = Top Plane

If there are existing faces or planes in the model, they can be chosen as sketching surfaces within the 3D sketch by using the **Ctrl** key while clicking the face/plane. A Face/Plane icon will then appear next to the cursor.

Planes can also be created *within* the 3D sketch to provide a planar surface to orient entities. **3D Sketch Planes** are sketch entities and are available on the **Sketch** toolbar. See *Creating a 3D Sketch Plane* on page 154 for more information.

Space Handle



Besides the cursor feedback, SOLIDWORKS provides another on screen tool to assist in keeping track of your orientation within the 3D sketch environment: the **Space Handle**. The **Space Handle** shows in red and its axes indicate the directions of the currently selected face or plane. The space handle follows the points that are placed within the 3D sketch to help you identify your orientation. Inference lines from the space handle show where automatic relations can be captured.

Subset of Sketch Entities and Relations

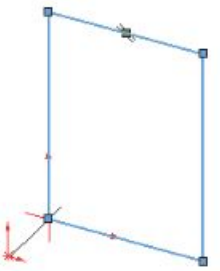
There are fewer entities and sketch relations available in 3D sketches compared to 2D sketches. When working in a 3D sketch, Vertical and Horizontal relations are replaced with the relations **Along X**, **Along Y**, and **Along Z**. Since there are more than just 2 dimensions available in the 3D sketch environment, these relations allow entities orientations to be fully defined by aligning them with the axes of the coordinate system of the model.

---

1 **New part.**
Create a new part using the Part_IN template. Name the part Frame.

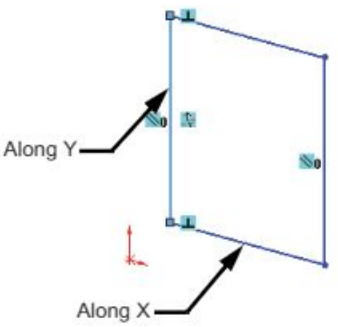
2 **Open a new 3D sketch.**
Click **3D Sketch** .

3 **Sketch rectangle.**
Using the default alignment to the Front Plane (**XY**), sketch a rectangle off the origin as shown.



4 **Relations.**
Click the left side vertical line to view the automatic sketch relations.

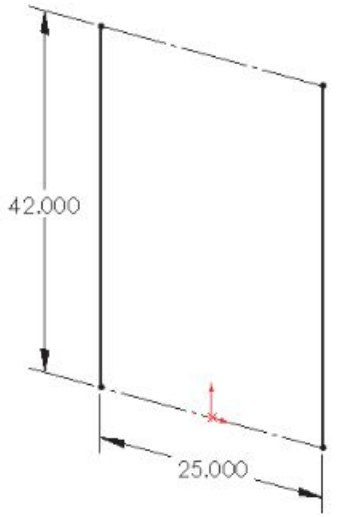
Add an **Along Y** relation to the left side vertical line and an **Along X** relation to the lower horizontal line.



5 Dimensions.
Change the horizontal lines to be **For Construction**.

Add a **Midpoint** relation between the lower horizontal line and the origin.

Add the dimensions shown.



Creating a 3D Sketch Plane

The **3D Sketch Plane** is a 2D plane positioned in a 3D Sketch. When active, sketch geometry conforms to the X and Y directions of that plane. The plane is defined using **First**, **Second,** and optionally **Third References** to geometry.

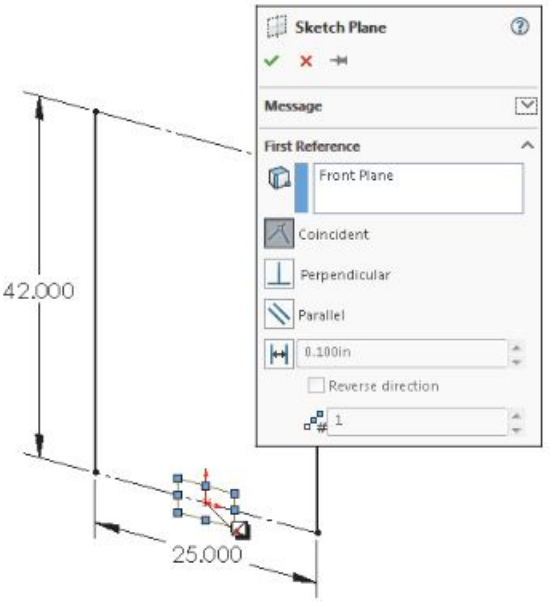
Where to Find It

- CommandManager: **Sketch > Plane**
- Menu: **Tools, Sketch Entities, Plane**

6 First reference.
Click **3D Sketch Plane**.

Select the Front Plane as the **First Reference**.

Click **Coincident** as the type.

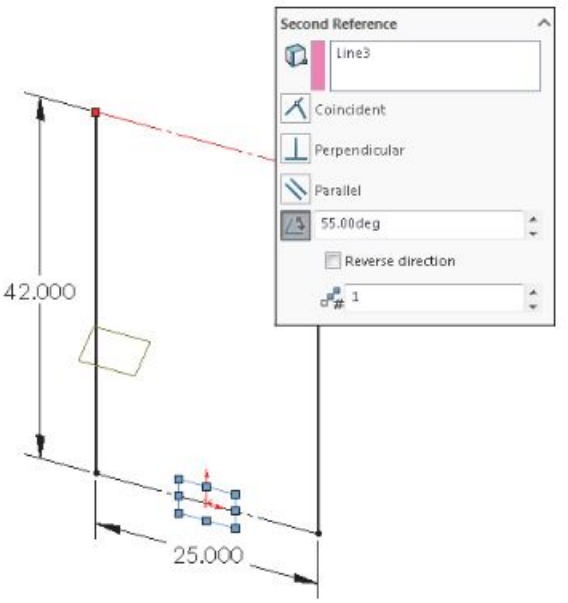


7 Second reference.
Select the upper construction line as the **Second Reference**.

Click the **Angle** type and set the value to **55°**.

Check **Reverse direction** if necessary.

Click **OK** ✔.



8 Active plane.
The plane shows a grid to indicate it is "active".

Active Planes

Active planes in a 3D Sketch appear with a grid across the plane preview. When a plane is active, entities added to the sketch will automatically be created with an **On Plane** relation.

When sketching on an active plane, only 2D sketch relations will be available.

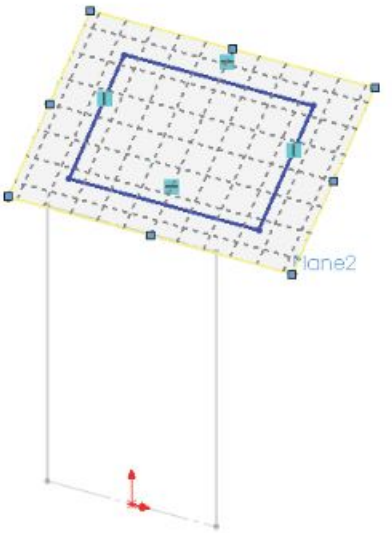
3D Sketch Planes will automatically become active after creation. Default reference planes and plane features can also be made active in a 3D Sketch. Planes are activated by double-clicking the plane preview. Planes are deactivated by double-clicking off of the plane preview.

---

9 **Rectangle.**
Sketch another rectangle above the existing geometry.

The 3D Sketch Plane will automatically resize to surround the geometry.

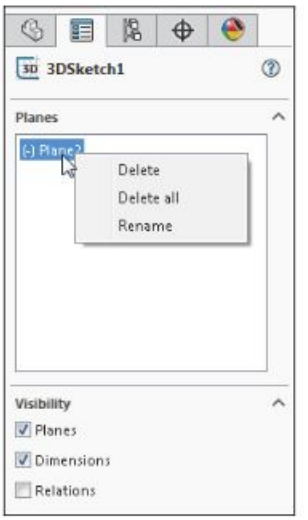
10 **Deactivate plane.**
Double-click off the plane to deactivate it. The plane appears light gray with dashed edges.



Visibility controls

The **3D Sketch** PropertyManager is available when nothing is selected in the sketch. Click the **PropertyManager** tab to access it. The top portion includes a list of **3D Sketch Planes** that have been created in the sketch. Right-clicking on the name allows you to **Delete**, **Delete all,** or **Rename** planes.

The **Visibility** controls below enable you to show or hide **Planes**, **Dimensions,** or **Relations** in the sketch.

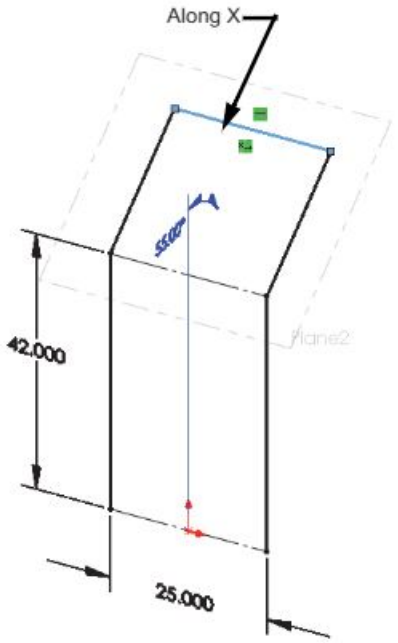


11 Relations.

Delete the lower line of the rectangle.

Use **Merge** relations to tie the endpoints to the existing geometry.

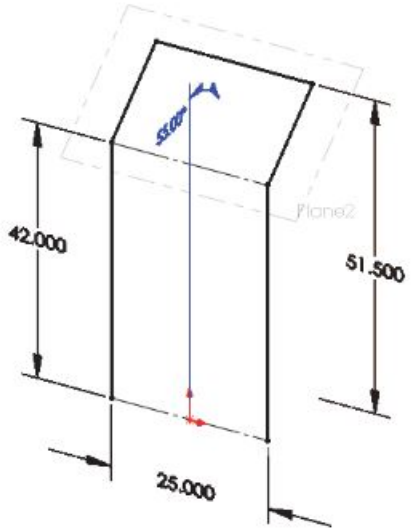
Add an **Along X** relation as shown.



12 Dimension to plane.

Create a dimension between the Top Plane and the upper line.

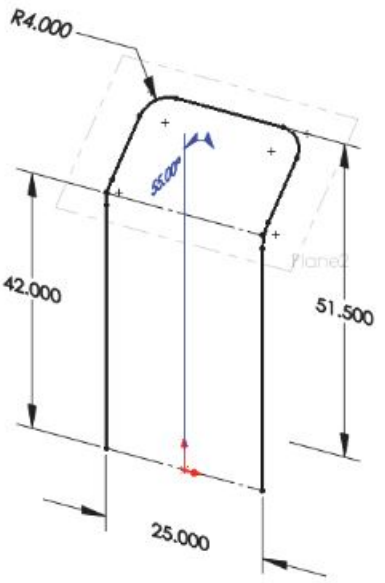
Set the value to **51.5"** to fully define the sketch.



13 Add fillets.

Add four sketch fillets of **R4"**. They are automatically set equal if created with the same command.

Adding these fillets would not be possible if these were 2 separate 2D sketches.



14 Exit the sketch.

15 Insert the structural member.

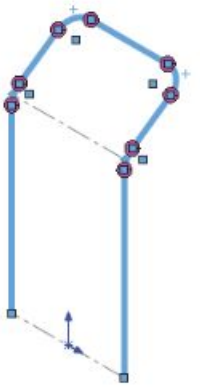
Click **Structural Member** .

Standard: ANSI Inch_Training

Type: Al Round Tubing

Size: 0.75 OD x 0.083 Wall

From the flyout FeatureManager, select 3DSketch1 to pick all entities of the sketch for the group.



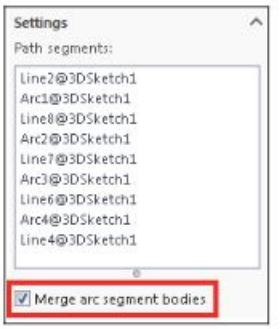
Merge Arc Segment Bodies

When the path consists of a series of lines and arcs, the option **Merge arc segment bodies** appears in the PropertyManager. Whether or not this option should be checked depends on how the weldment will be fabricated.

If the weldment is fabricated using bent tubing, the option should be checked. This will result in a model where lines and tangent arcs are merged into a single solid body.

If the weldment is fabricated using elbows and straight tubes, the option should be cleared. This will result in a model where each straight segment is a separate body and each arc segment (elbow) is a separate body.

16 Merge arc segment bodies.
Select the **Merge arc segment bodies** checkbox.



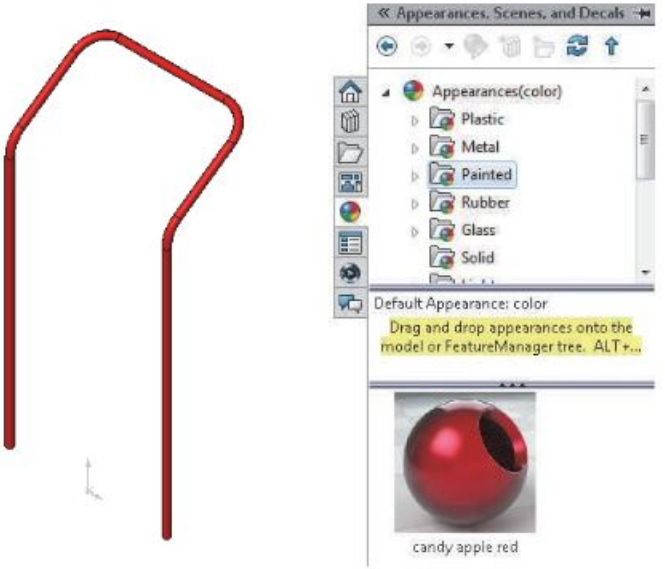
Click **OK** ✔ to create the structural member.

17 Results.
The structural member is represented as a single solid body.

There should only be 1 solid body appearing in the Cut List.

The cut list item properties include a length property that reflects the centerline length of the bent tube.

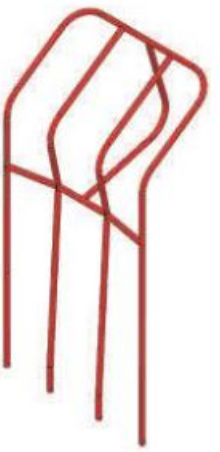
18 Modify appearance.
Use the Task Pane to apply Painted appearance of candy apple red to the Frame.



Next Step

The remaining sketch layouts can be created as 2D Sketches, but planes will need to be created to properly locate them.

Since the additional members of the frame share the same profile, they can be created as new groups within the same structural member feature. To accomplish this, rollback the FeatureManager design tree to add the layout sketches before the feature.



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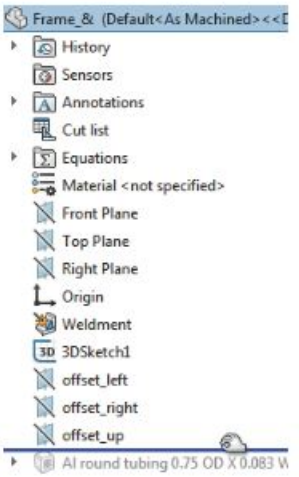
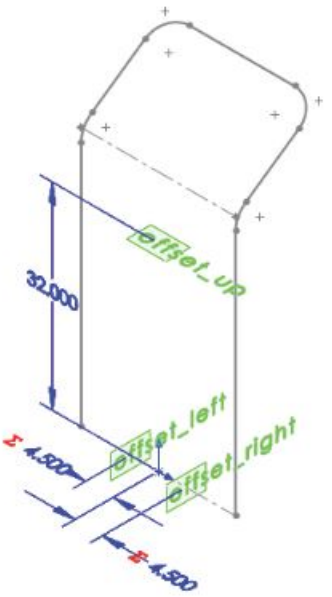
19 Rollback.

Position the rollback bar before the structural member feature in the FeatureManager design tree.



20 Reference planes.

Create three reference planes as shown in the illustration at the right. Notice the use of a global variable to link `offset_left` and `offset_right`.



Tip

Press **Ctrl** while dragging a plane preview as a shortcut for creating an offset plane.

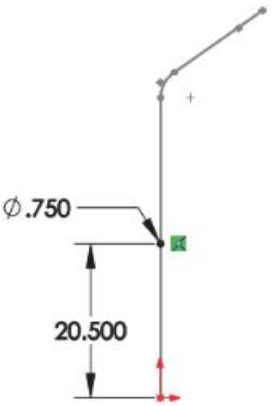
21 2D sketch.

Select `offset_left` and open a **Sketch**.

Change to the **Right** view.

22 Set up for clearance.

Sketch a **0.750"** diameter circle and position it as shown. The center of the circle is **Coincident** with the vertical line in the 3D sketch.



Why?

The hand truck has a movable platform (the fold nose extension) that folds up as shown below. The circle will be used to determine the angle of the structural member so it is bent back far enough to provide clearance when the platform is in the up position.

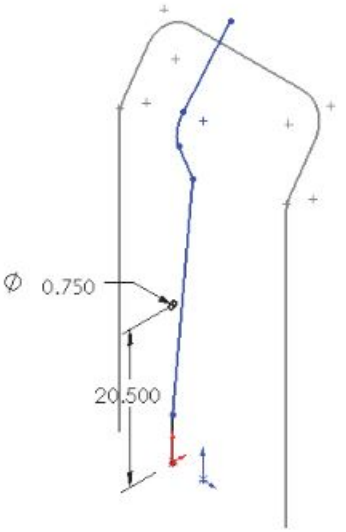
Extension in down position

Extension in up position

23 Continue sketching.

Starting at the sketch origin, sketch a vertical line approximately **5"** long.

Then sketch the remaining lines and tangent arc as shown.



24 Add relations.

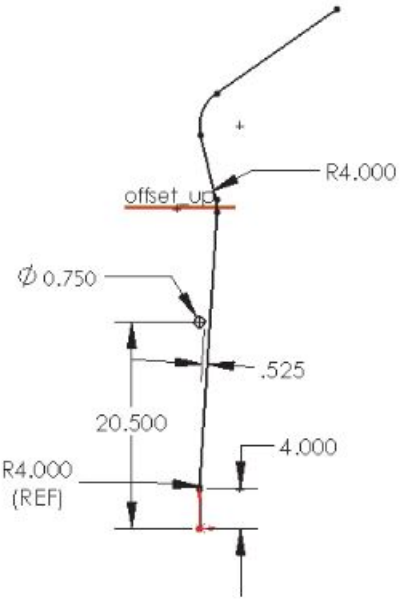
Add a **Coradial** relation between the tangent arc and the **4"** fillet in the 3D sketch.

Add a **Pierce** relation between the end of the angled line and the horizontal line in the 3D sketch.

25 Fully define the sketch.

Add a **Coincident** relation between the uppermost sharp corner and offset_up.

Add dimensions and fillets as shown.



26 Exit the sketch.

27 Derived sketch.

Select the 2D sketch and offset_right.

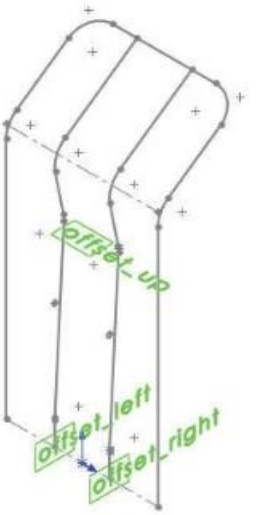
Click **Insert, Derived Sketch**.

28 Fully define the sketch.

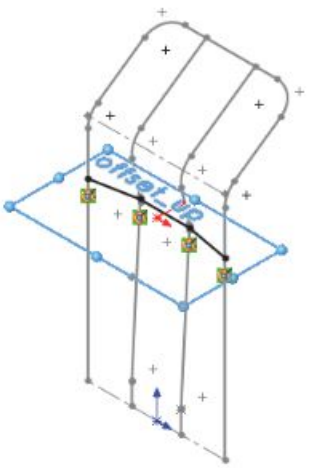
Add a **Coincident** relation between the lowermost end of the vertical line and the origin.

Add a **Collinear** relation between the vertical lines in the sketches.

29 Exit the sketch.



30 New 2D sketch.
Select the plane offset_up and open a 2D sketch.



Sketch three lines as shown. The ends of the lines are constrained to the 3D sketch and the *arcs* in the two 2D sketches using **Pierce** relations.

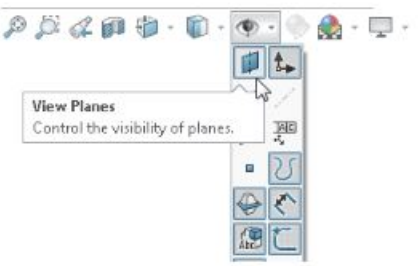
Tip

Pay attention to the cursor feedback to identify when an arc is being selected.

31 Exit the sketch.

32 Optional: Hide the planes.
Visibility of planes can be controlled individually by selecting them and clicking **Hide** or **Show**.

Alternately, use the **Hide/Show Items** menu to control the visibility of all planes, regardless of their Hide/Show state.



33 Edit feature.
Move the rollback bar to position it after the structural member feature.

Edit the structural member feature.

34 New Group.
Click **New Group**.

Select Sketch2 from the flyout FeatureManager.

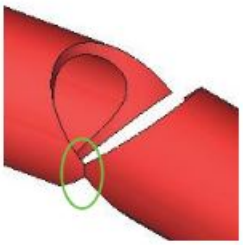
Make sure **Merge arc segment bodies** is selected. All other default settings are correct.

35 New Group.
Repeat step **34** to create a new group for the derived sketch.

36 New Group.
The final group will be separate pieces of tubing.

Apply a corner treatment of **End Miter** between the members.

There is a very slight overlap where these members meet. Using the corner treatment will ensure they are properly trimmed.



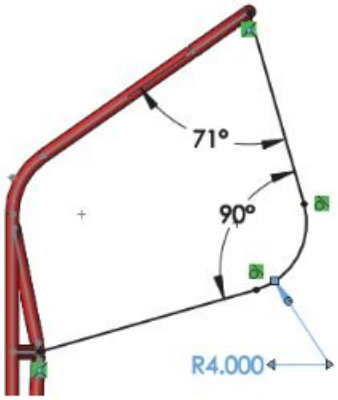
Click **OK** ✔.

Tip

Image Quality settings may need to be adjusted to better view the results of the automatic trimming in the part. These settings are found in **Options**, **Document Properties, Image Quality.**

37 New 2D sketch.
Select the Right Plane and create the sketch shown in the illustration at the right.

Exit the sketch.



38 Reorder.
To use this sketch as a new group in the existing structural member feature, reorder it so it precedes the feature in the tree.

39 Edit feature.
Edit the structural member feature.

Add a **New Group**. Select the sketch from the flyout FeatureManager or individually select the sketch segments.

Merge arc segment bodies.

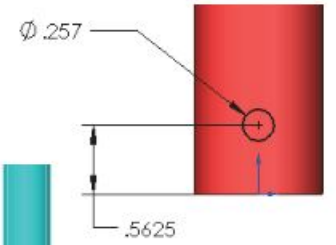
Click **OK** ✔.

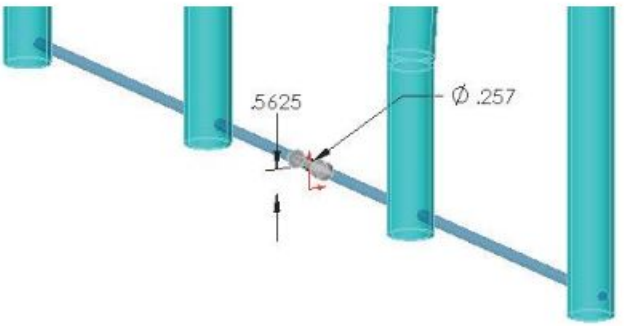
Optional: Finishing Details

Complete the frame by adding the finishing details in the remaining steps.

40 Pivot hole.

Create a sketch on the Right Plane as shown. Extrude a **Through All** cut in both directions.

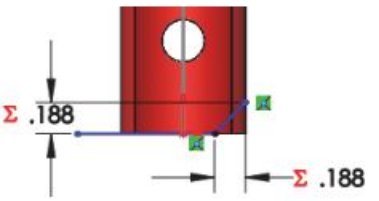




41 Chamfer.

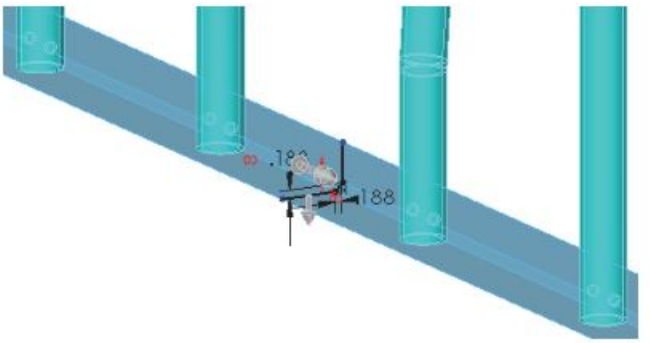
Create a sketch on the Right Plane as shown. Extrude a **Through All** cut in both directions.

Use **Flip side to cut** if necessary.



Note

The solid bodies were made transparent for illustration purposes.

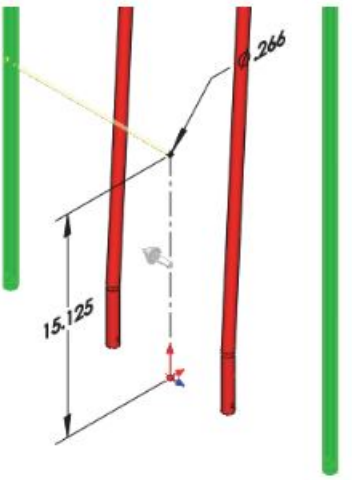


42 Hole for retaining pin.
Create a sketch on the Right Plane as shown.

Extrude a cut **Through All** in **Direction 1**.

Use **Feature Scope** to only affect the perimeter body.

43 Save the part.



Exercise 10: Chair Frame

Create a chair frame produced from pieces of bent tubing.

This lab reinforces the following skills:

- *3D Sketching* on page 152
- *Creating a 3D Sketch Plane* on page 154
- *Merge Arc Segment Bodies* on page 158
- *Trim/Extend Options* on page 26

Units: **mm**

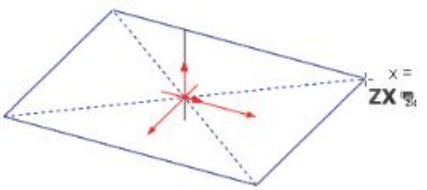
Procedure

1 New part.
Create a new part with the `Part_MM` template.

2 Open a new 3D sketch.
Click **3D Sketch**.

3 Sketch rectangle.
Click **Center Rectangle**.

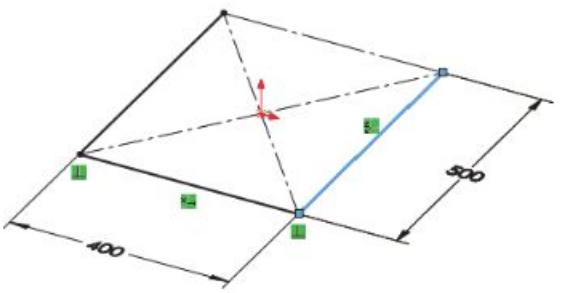
Press **Tab** until orientated to the `Top Plane` (**XZ**).



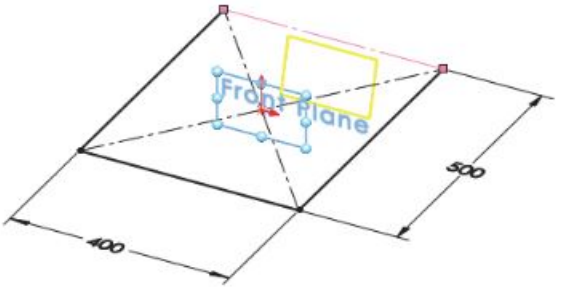
Create the rectangle centered on the `Origin`. Change the back line segment to be **Construction Geometry**.

4 Add relations and dimensions.
Add **Along X** and **Along Z** relations.

Dimension the rectangle to be **400mm** by **500mm**.



5 3D Sketch Plane. Create a **3D Sketch Plane** with the following references:



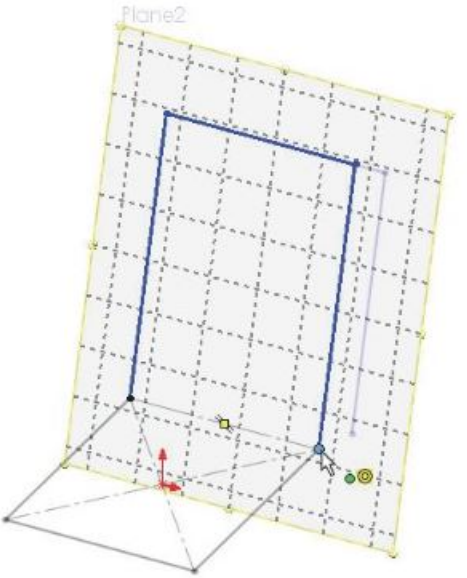
First Reference: Front Plane

Second Reference: Construction Line

Angle: 15.00 deg, Reverse direction

Click **OK** ✔.

6 Sketch Corner Rectangle. Create a **Corner Rectangle** on the active plane.



Delete the lower line segment of the rectangle.

Drag the endpoints to **Merge** with endpoints in the first rectangle.

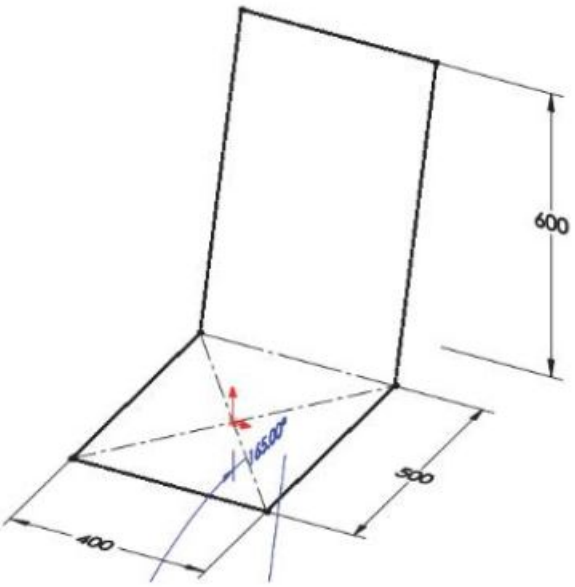
Double-click outside of the plane preview to deactivate the plane.

Note

In the remainder of the lesson illustrations, the 3D Sketch Plane has been hidden. To control the visibility of 3D Sketch Planes, see *Visibility controls* on page 156.

7 Dimension to Top Plane.

Add a **Dimension** between the upper line and the Top Plane, **600mm**.

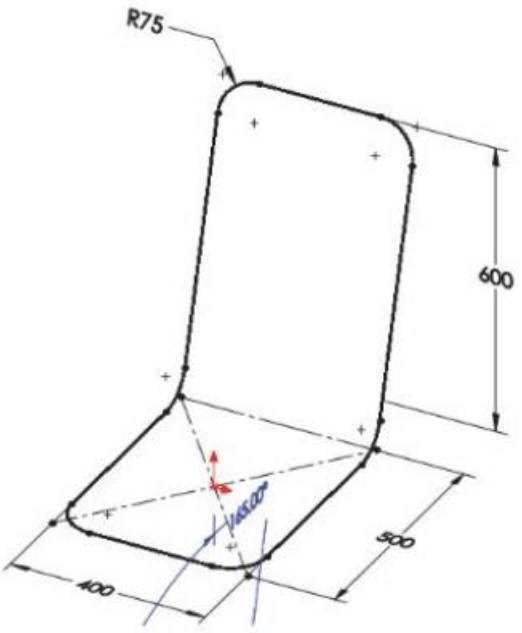


8 Add fillets.

Add **75mm** fillets to the sharp corners of the sketch.

Note

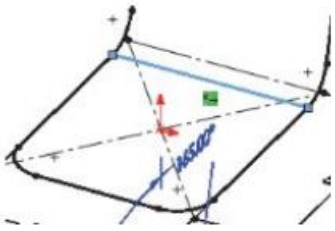
Adding these fillets in multiple directions would not be possible with 2D sketches.



9 Add a line.

Sketch an additional line between the arc endpoints as shown.

10 Exit the sketch.



11 Create structural members.

Click **Structural Member** .

Use the following settings to create the bent piece for the frame.

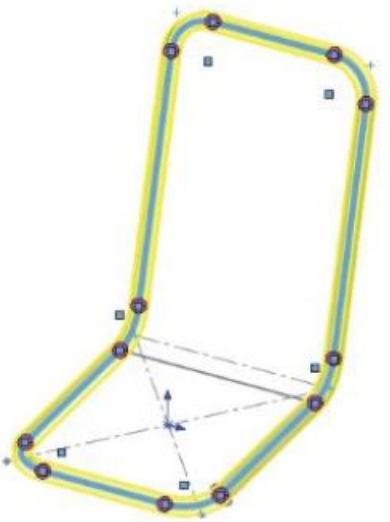
Standard: ISO_Training

Type: Tube (square)

Size: 30 x 30 x 2.6

Select the sketch segments shown as Group1.

Select **Merge arc segment bodies**.



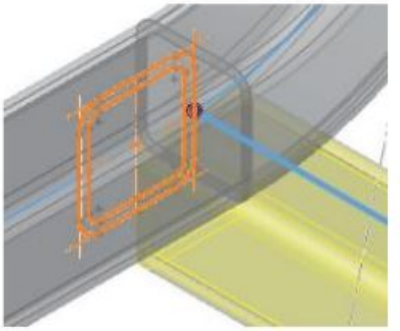
12 New Group.

Click **New Group**.

Pick the remaining sketch segment as Group2.

Click **Locate Profile**. Click the point as shown.

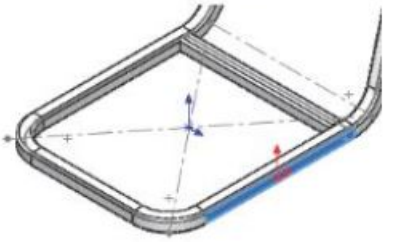
Click **OK** .

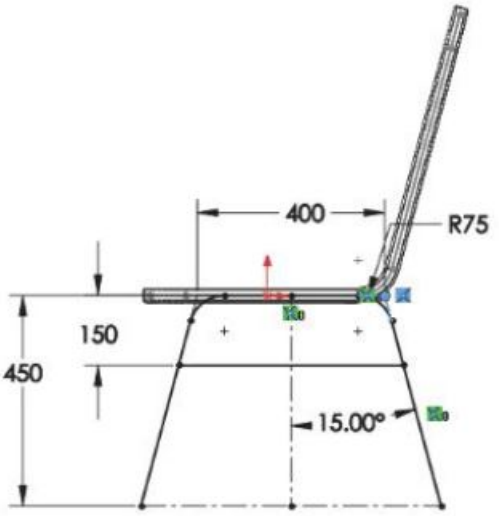


13 Sketch Leg Layout.

Start a sketch on the face highlighted at the right.

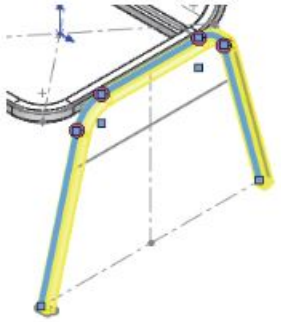
Create lines and tangent arcs to layout the legs as shown.





14 Exit the sketch.
Exit the sketch and name it Leg Layout.

15 Create structural members.



Click **Structural Member** .

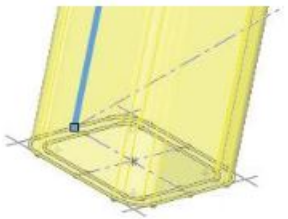
Use the following settings to create the bent piece for the frame.

Standard: ISO_Training

Type: Tube (square)

Size: 30 x 30 x 2.6

Select the sketch segments shown as Group1.



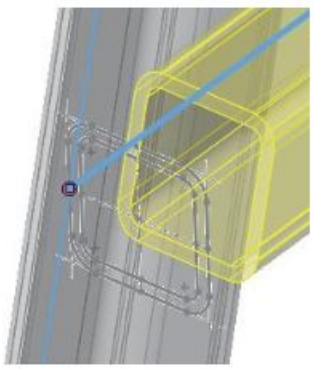
Select **Merge arc segment bodies**.

Click **Locate Profile**. Click the point as shown.

Note

Profile position will vary based on how the sketch entities are created and which segment is chosen first.

16 New Group.
Click **New Group**.



Select the remaining sketch segment as Group2.

Click **Locate Profile**. Click the point as shown.

Click **OK** .

17 Create a plane for trimming.
The bottom of the legs will need to be trimmed to be flush with the ground.

Create a **Plane** that is **Parallel** to the Top Plane and **Coincident** with a point in the Leg Layout sketch.

Rename the plane Bottom of Legs.

18 Trim/Extend.

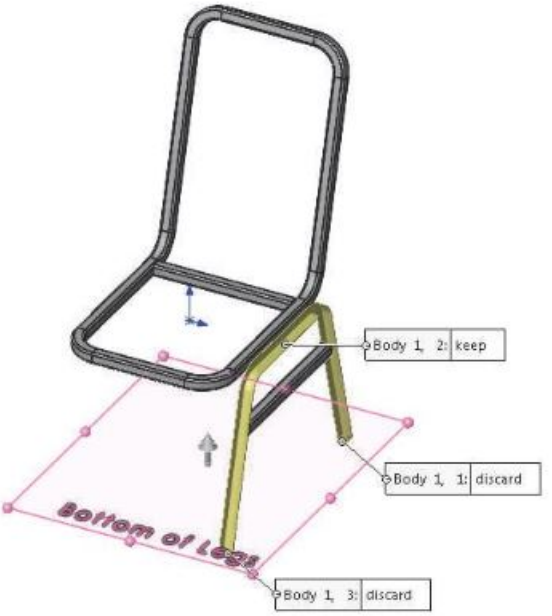
Hide the sketches.

Click **Trim/Extend** .

Select the bent leg body as the **Bodies to be Trimmed**.

Select the Bottom of Legs plane as the **Trimming Boundary**.

Click the flags in the graphics area to toggle the areas to **Keep** and to **Discard** as appropriate.



19 Mirror bodies.

Mirror the leg bodies across the Right Plane.

20 Optional: Review the Cut List.
Use the **Cut List Properties** dialog to review the cut list items and properties.

21 Save and close all files.

Exercise 11: Bent Tubing, Sheet Metal, and Assemblies

Create this set of movable warehouse steps by following the procedure given.

This lab reinforces the following skills:

- *Working with Bent Structural Members* on page 152

Units: **Inches**

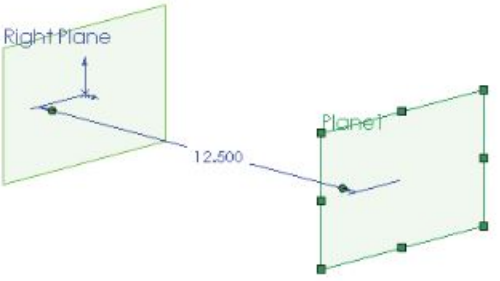
Procedure

The first part created will be the frame for the stair tread. Build one side frame, mirror it, and then add the structural members that connect the two sides.

1 New part.
Create a new part using the Part_IN template.

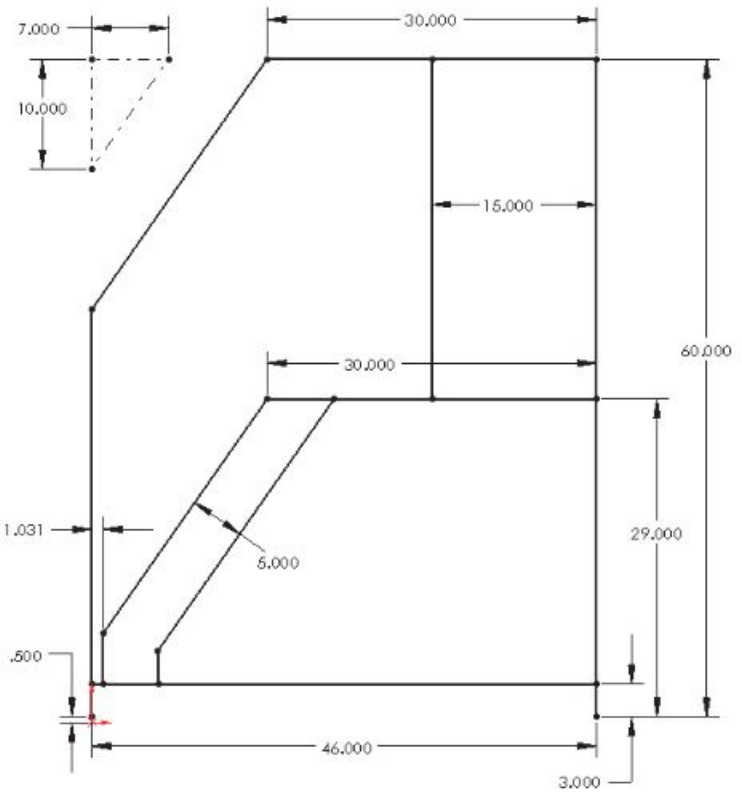
Name the part Stair_Frame.

2 Reference plane.
Create a reference plane **12.5"** from the Right Plane as shown.



3 Sketch.

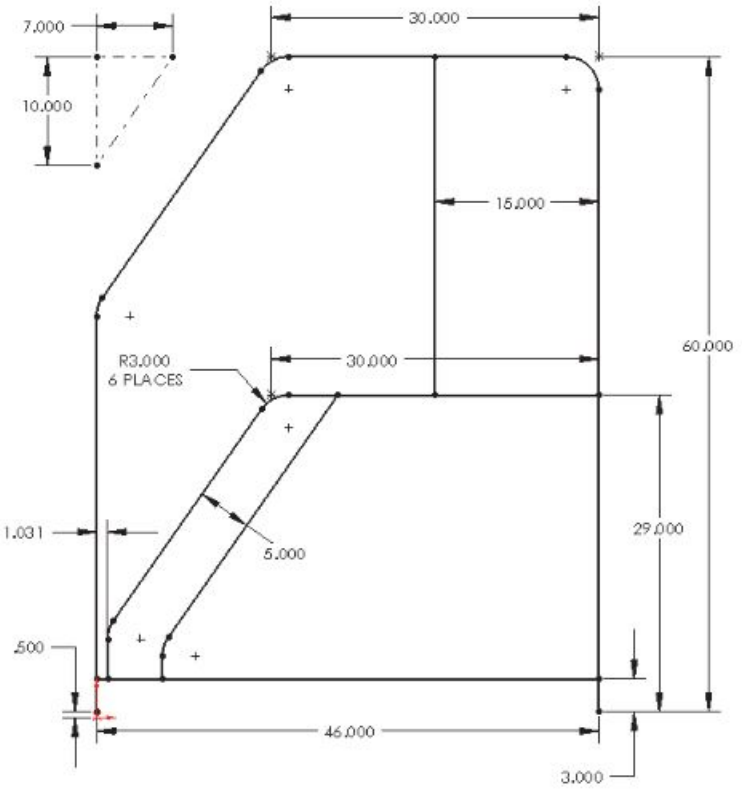
Create a sketch on `Plane1` as shown below.



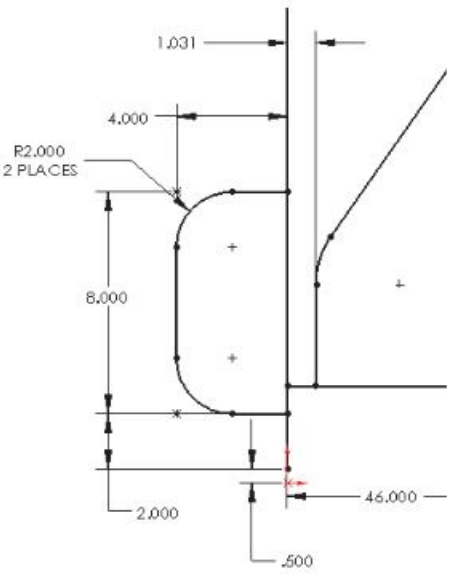
Tip Stairs are usually specified by their rise and run rather than by some odd angle dimension. An easy way to do this is to sketch a triangle using construction geometry and dimension the rise and run. Then make the stringer of the stairs parallel to the hypotenuse of the triangle.

Note Some information may not be present in the sketch images and may need to be inferred or assumed.

4 Sketch fillets.
Add six sketch fillets with a radius of **3"** as shown below.



5 Finish sketching.
Complete the sketch by adding the geometry as shown at the right.



Note

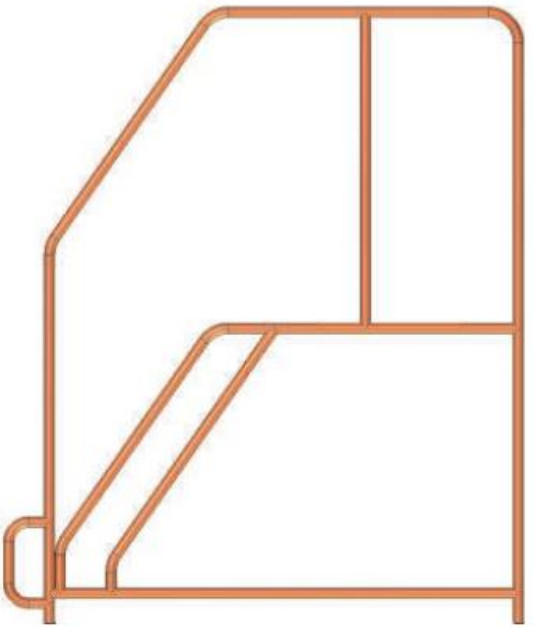
The sketch fillets are **2"**, not **3"** as they were in step **4**.

6 Exit the sketch.
Name the sketch Frame Sketch.

7 Structural members.

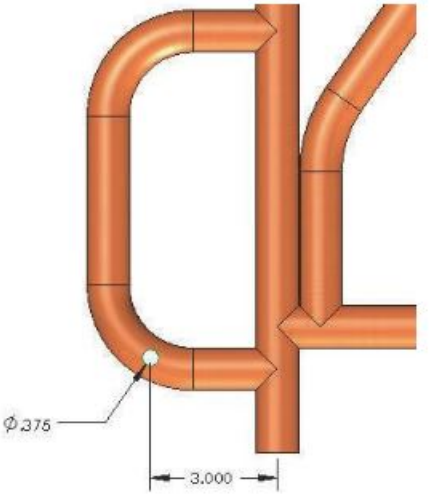
Insert structural members using ANSI Inch_Training, AI Round Tubing, 1 OD x 0.095 Wall seamless tube as shown below. This will require several separate groups.

All arc segments should be merged to produce bent tubing.



8 Extrude a cut.

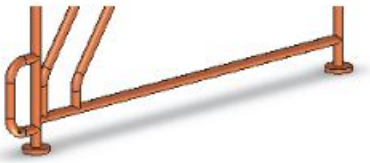
Select Plane1 and open a sketch. Sketch a **0.375"** circle as shown. Extrude a cut using the end condition **Through All** in both directions.

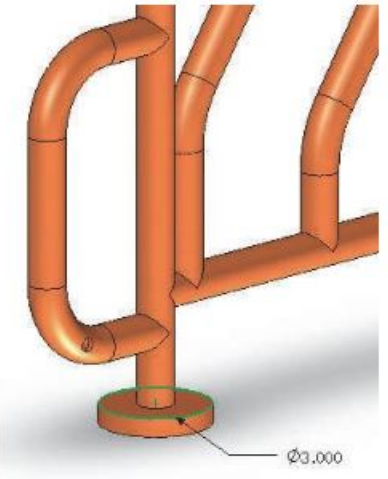


9 Add the foot pads.

Select the bottom face of the vertical structural member and open a sketch. Sketch a **3"** diameter circle concentric with the bottom end of each vertical segment.

Extrude the sketch **0.5"**.

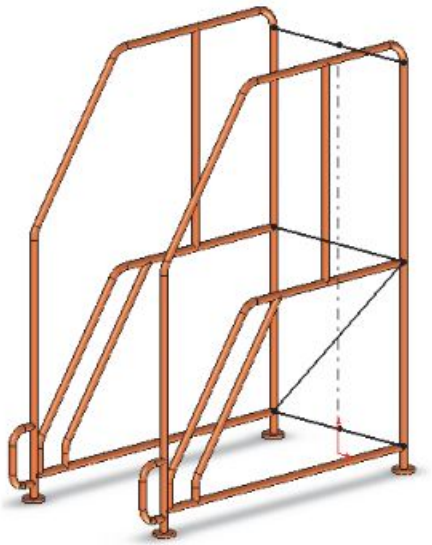




10 Mirror the side frame.

Select the Right Plane and mirror *all* the bodies to create the second side frame.

11 Sketch the path segments for the crosspieces.



You can sketch this as a 3D or 2D sketch. One technique using a 2D sketch is to:

1. Create a reference plane parallel to the Front Plane and coincident with one of the endpoints of the Frame Sketch. This way if the dimensions of the Frame change, the position of the reference plane will update.
2. Select this new plane an open a 2D sketch.
3. Sketch a vertical centerline and make it coincident with the Origin.
4. Sketch three lines for the horizontal path segments. Make their starting points coincident with the ends of the path segments in the Frame Sketch.
5. Make the midpoints of the three lines coincident with the centerline.
6. Finally, sketch the diagonal path segment.

12 Create the structural members and trim them.

Create and trim the structural members as necessary.

13 Save the part.

14 Create an assembly.

Click **Make Assembly from Part/Assembly** and select the Assembly_IN template.

In the **Part/Assembly to Insert** list, make sure the Stair_Frame is selected.

Click **OK** to insert the component at the origin.

15 Save the assembly.

Name the file Movable Steps.

Adding the Stair Tread

The next part added to the assembly will be a sheet metal tread plate. The Tread part has two sizes: one 10 inches deep and one 7 inches deep. You can choose to design the tread part yourself, or use the existing part called Tread in the Lesson04\Exercises folder.

Optional: Designing the Tread

If you chose to design the tread, use the existing part as a guide to create the sheet metal features and necessary configurations: 7x24 and 10x24. Each configuration has an optional derived configuration which includes the counter sink emboss feature and pattern for the non-slip surface shown below.

---

16 Insert a component.

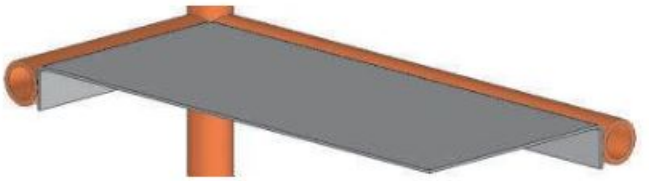
Click **Insert Components**. Add the Tread component to the assembly.

Use the 10x24 configuration.

17 Mate component.

Use the section view below as a guide to mate the Tread as follows:

- The uppermost face of the Tread is **Tangent** to the cylindrical face of the middle crosspiece.
- The rear face of the Tread is **Tangent** to the cylindrical face of the crosspiece.
- The uppermost face of the Tread is **Parallel** to the Top reference plane of the assembly.
- The Right Plane of the Tread is **Coincident** with the Right Plane of the assembly.



18 Insert two more copies of the Tread.

These instances also use the 10x24 configuration.

Mate the treads so they are fully defined and positioned as shown in the illustration at the right.

19 Add the components for the steps.

These instances use the 7x24 configuration.

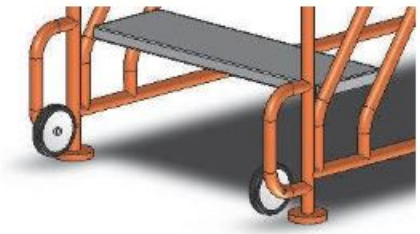
Mate the components as follows:

- Use a **Distance** mate of **10"** to control the spacing between the uppermost faces of the Tread components.
- The rear face of the Tread is **Coincident** to the front face of the Tread above it.
- The Right Plane of the Tread is **Coincident** with the Right Plane of the assembly.

20 Add the wheels.

Add two instances of the Wheel - 4 in component.

Mate them **Concentric** to the holes and **Tangent** to the Frame as shown in the illustration.



21 Results.

Optional

- Create a configuration of the assembly which uses the derived Tread configurations with the non-slip surface shown.
- Generate the cut list for the Stair_Frame part.
- Make a drawing of the Stair_Frame and insert a cut list table.
- Make an assembly drawing of the Movable Steps complete with a bill of materials.

22 Save and close all the files.

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